Machine-Shop Practice

HARRY A. JONES

Book II

PROGRESSIVE LESSONS IN

MACHINE SHOP PRACTICE

BOOK II

BY
HARRY A. JONES
Machine Shop Instructor,
Technical School,
Vancouver, B.C.

THE BRUCE PUBLISHING COMPANY
NEW YORK MILWAUKEE CHICAGO

Reprinted 1931, 1937, 1939, 1940, 1941, 1942

PRINTED IN CANADA

T. H. BEST PRINTING CO., LIMITED
TORONTO, ONT.

PREFACE

After assimilating the knowledge presented in Book I, the student will find that the work in this book is on a little higher level. However, with the foundation already made with the first book, he should have no difficulty in understanding the work presented here.

It is well to realize that Books I and II do not by any means cover the limits of this field of work, but do give basic, fundamental lessons, which I have found to be important after many years of instructing in Technical Schools. Outstanding difficulties of the average students have been noted, and these lessons are intended to ease them over important parts of their course when misunderstanding is likely to occur.

The student will find that after covering the work in Books I and II, he can read intelligently and study the many excellent books of the chapter type, and the up-to-date bulletins so freely offered by the many industrial firms. It has also been found, from experience with students, that definite lessons such as are presented here, supply their immediate needs, and can be assimilated in the least possible time. The student need not spend his time writing notes, and when revision time for examination comes, his allotment can be specified in a definite manner.

Books I and II, have been written on the assumption that there are "levels of learning", and that the student learns best when presented with work he is able to master, and is not worried by difficult things until he is well able to learn them. The writer feels sure that after this work has been covered thoroughly, the student will find that he can confidently and independently use the many excellent books available in the more advanced sections of this field of work.

HARRY A. JONES.

Vancouver, B.C.,
July 11, 1930.

ACKNOWLEDGMENTS

The author and the publishers wish to acknowledge with thanks the valuable assistance given by the following gentlemen:

Mr. John Kyle, Director of Technical Education for the Province of British Columbia.

Mr. J. G. Lister, Principal of the Vancouver Technical School of Vancouver, British Columbia.

CONTENTS

Section I—Benchwork

Section II—Drilling

Section III—Lathework

Section IV—Planing in the Shaper

Section V—Milling

Section VI—Grinding

Section VII—Shop Mathematics

Section VIII—Shop Science

Tables

BENCHWORK

BABBITTING BEARINGS

Babbitt is an alloy metal, usually consisting of varying proportions of tin, antimony, lead and copper, with sometimes a trace of arsenic or bismuth. Babbitt is used as a means of lining a bearing because of its anti-friction properties. When the bearing becomes worn, the babbitt can be melted out and the bearing re-babbitted. Although jigs are used in standard work, it is advisable for a bench hand to be able to pour a solid or split type of bearing with equipment on hand.

Solid type of bearing. Diagram (1). Before babbitting such a bearing, provision must be made to support the mandrel in the correct axial position so that the bearing when poured will be in correct alignment with the rest of the machine. Stops must be used to prevent the molten babbitt from leaking out. If clay or putty is used in the manner shown on the left of diagram (1), the babbitt will not finish square with the end of the bearing. A piece of cardboard, as shown in diagram (2), should be used to stop the end square and this can be supported and sealed by clay or putty or by a wood or metal stop on the outside.

Air vents. Diagram (3). All bearings should have an air vent so that, as the molten metal rushes into the bearing, the air will not be trapped but can escape freely. *Pouring dams* will provide a means of pouring the metal fast. It is very important in pouring a bearing, that sufficient metal is available and the metal should be poured continuously to obtain a homogeneous babbitt metal layer. A poor job is shown in diagram (3). A joint shows where the pouring stopped and if tested the upper part would separate easily from the lower part.

Heating the bearing before pouring. If at all possible, this precaution is essential so that the soft metal will not chill and solidify before the bearing box is full. Heating the bearing is also necessary from a safety standpoint because if there is moisture in the bearing there will be an explosion.

Heating the babbitt. The babbitt should be just hot enough to char a thin pine stick which is immersed in the babbitt to test the temperature. It is advisable to stir the babbitt before pouring, to be sure that all the metals are properly mixed. The dross which forms on top of the molten metal in the ladle should be skimmed off just before pouring. A small piece of rosin added just before pouring helps to make a smoother bearing. If the babbitt is overheated the various metals tend to separate

and some lose their anti-friction property. If the babbitt remains heated for any length of time, keep it covered with a layer of broken charcoal to prevent undue oxidation.

Anchoring the babbitt to the bearing. Some bearings have dovetail grooves cast in them, as shown in the two views in diagrams (4) and (5). Others have holes drilled in them, as shown in diagrams (7) and (8). Either of these methods locks the babbitt to the bearing and can be improved by peening the babbitt with a hammer to make the babbitt adhere. Without some anchor, the babbitt would loosen and turn with the shaft it is intended to support.

Pouring a split bearing, as in diagrams (6) and (7). Cardboard liners are used to provide spaces for the shims, and these liners are cut as shown in diagram (6) to allow the babbitt to run to the lower bearing when being poured. Large bearings are often poured in a vertical position to prevent shrinkage holes forming in the babbitt as it cools.

Properties of the metals used in babbitt. *Lead* is a good anti-friction metal but it is very soft.

Tin gives strength, toughness and makes the metal dense. It also acts as a solder to unite the metals.

Antimony gives hardness and expands when cooling, which helps to lock the babbitt in the anchor holes.

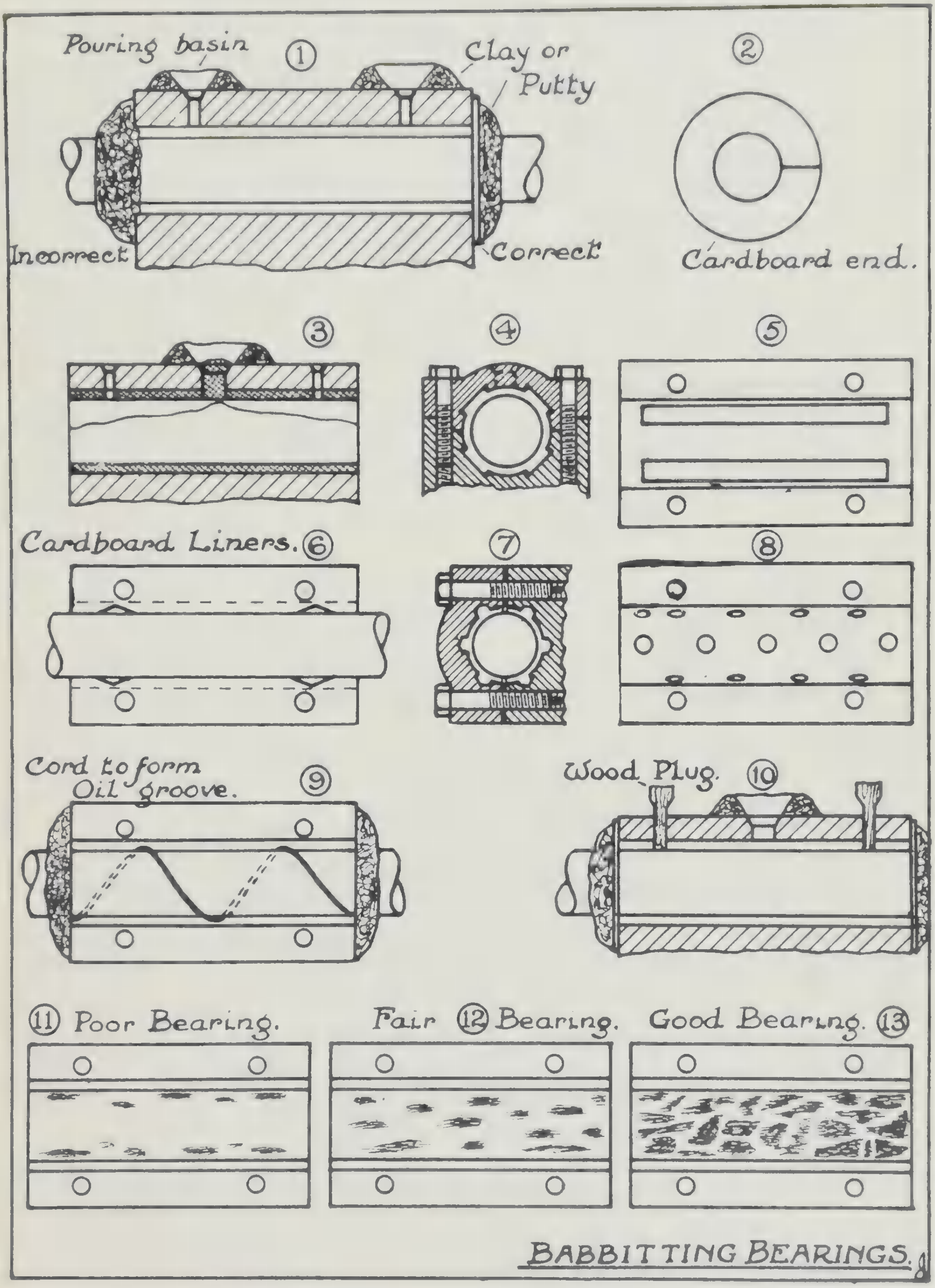
Copper creates toughness and hardness.

Bismuth is sometimes used to prevent shrinkage.

Forming oil grooves. One method is shown in diagram (9) by wrapping a cord around the mandrel before pouring. Usually the oil grooves for distributing oil throughout the bearings are cut with a round-nosed bent chisel. *Oil holes* can be left open by placing wooden plugs in them before pouring the babbitt, as shown in diagram (10).

Removing the Mandrel. This must be done as soon as the metal sets, as the contraction of the babbitt will hold the arbor or mandrel tightly and make it difficult to remove. A piece of paper may be wrapped around the shaft before pouring; this reduces the amount of scraping and allows one to remove the mandrel with ease. A coating of cylinder oil on the mandrel also helps to make its removal easier.

Scraping the bearing. A shaft is used in the bearing to test the accuracy of the scraping. The shaft is covered with prussian blue to mark the high spots, as shown in diagrams (11), (12), (13). A bearing scraped as shown in (13) is considered good.


Pouring basin
1
Clay or Putty
2
Incorrect
Correct
Cardboard end.
3
4
5
Cardboard Liners. 6
7
8
Cord to form Oil groove.
9
Wood Plug.
10
11 Poor Bearing.
Fair 12 Bearing.
Good Bearing. 13
BABBITTING BEARINGS.

BELT FASTENINGS

There are many methods used to fasten the ends of belts together, but the most effective method is by shaving down the ends of the belt to form a "scarf" and cementing the two parts together. From a safety point of view, this is unquestionably the best method to consider. Many mechanics have received bad cuts from moving by hand belts fastened with metal fastenings. Moreover, whenever a belt is pierced for metal fastenings or for leather lacing, the belt is very much weakened and will likely break at the weakened part. On grinding machines it is easy to notice the advantage of the cemented joint over all other methods, as it forms a continuous belt, whereas if metal fastenings are used, they give a distinct knock each time they pass over the pulley of the machine.

Direction to run the belt. Single belts should run with the grain or hair side next to the pulley, with the laps pointing in the direction shown in Diagram (1). This prevents the point of the lap on the inside coming undone, as it might if it worked in the opposite direction against the pulley pressure.

Diagram (2) shows a double belt which can only run with the grain side next to the pulley, as the hair sides are cemented together. The direction of the belt is shown running with the inside scarf in the same direction as the single belt scarf.

Lacing a leather belt. One method of lacing a leather belt with leather lacing is illustrated in diagrams (3) and (4).

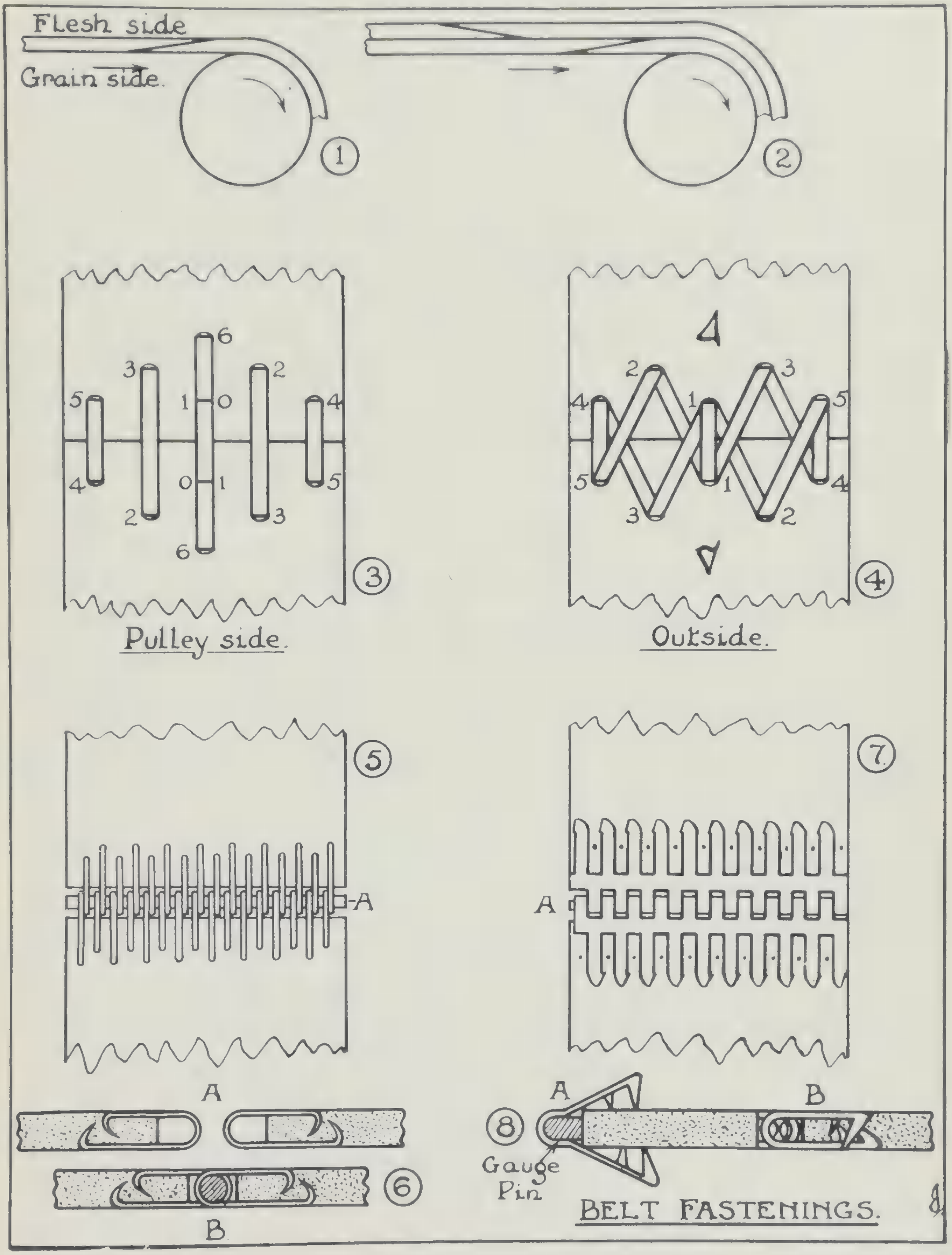
Diagram (3) shows the grain and pulley side of the belt and diagram (4) the reverse or flesh side of the belt.

There are many different methods of belt lacing used. The one illustrated here gives the minimum tendency to break the belt, due to the punching of the holes which weakens the belt. The holes are staggered out of line to reduce breakage to a minimum.

For 1 inch to 2 inch belts, $\frac{1}{4}$ inch lacing is sufficient; and three holes in each side of the joint are sufficient. For 2 inch to $4\frac{1}{2}$ inch belts, 3 to 5 holes at each side are sufficient. Up to 6 inch width of belt $\frac{3}{8}$ inch lacing is strong enough.

The holes should be punched a distance of $\frac{1}{2}$ inch from the end, for small belts the second row being 1 inch from the end. Larger belts are punched a greater distance from the end in proportion to the size of the belt.

Method of lacing. Start at the centre and work outwards, then inward to the centre. Start at 1, 1 in the centre with each side of the



BELT FASTENINGS.

leather lacing equal in length. Lace outwards on one side 1 to 2, 2 to 3, 3 to 4, 4 to 5, 5 to 4, 4 to 5. Inwards 5 to 2, 2 to 3, 3 to 1, 1 to 1, 1 to 1, 1 to 6 and out.

Repeat this operation on the other side and cut a short slit on the end of the lace to form a hook to prevent it from coming out.

Metallic belt fasteners. There are many belt fasteners on the market and the one selected should be one that will give a smooth, strong, flexible joint.

The clipper belt hooks are illustrated in diagrams 5 and 6. They are supplied fastened to cardboard to the correct spacing and a length is cut off the card with a knife to suit the width of the belt. The hooks are arranged alternately long and short and are fastened to the end of the belt usually with a special machine made for that purpose. Diagram (6A) shows the two ends of the belt with the hooks attached. Diagram (6B) shows the belt hooks interlocked with the rawhide hinge pin (A) diagram (5) in place. Care must be taken to see that the belt is held square with the machine while the hooks are being attached.

Alligator steel belt lacing is illustrated in diagrams 7 and 8. The main feature of this lacing is its easy, rapid application without any other equipment than a hammer.

A gauge pin is sometimes used to provide sufficient room for the hinge pins (B) diagram (8), which rock on each other as the belt rotates around the pulley.

Diagram (8A) shows the lacing set ready for hammering, and diagram (8B) the lacing after hammering; the points are riveted over on the opposite side of the belt making the surface smooth. The dots in diagram (7) show the points after riveting with the hammer.

Cutting the belt to length. Measure for the length of belt required by placing a steel tape around the pulleys; then deduct from this length $\frac{1}{8}$ inch per foot to allow for stretch. If the belt is to be cemented, allow the proper amount in addition to this length for the laps. The belt should be cut absolutely square with a square and a knife.

TYPES OF CHISELS AND THEIR USES

The cold or chipping chisel may be placed next in importance to the file as the most useful among the hand tools. In the days before the use of the shaper, planer, and milling machine, the chisel was a much-used tool. With the introduction of pneumatic hammers, the chisel has had its capacity for work increased and to-day it performs many operations where it is impossible to use other machines.

Chipping cast iron. Diagram (1) shows a section of a piece of cast iron. The outer surface of the metal has a very hard skin which resists penetration. If the chisel were used on this skin by taking a shallow cut, it would quickly be dulled. The weakest part of a chisel is its edge and this should be always beneath the skin, cutting in the soft metal. Diagram (A) shows the plan of the cold chisel showing the edge slightly convex looking at the broad face of the chisel. The corners of the chisel are naturally weak because they have no metal support on one side. If the edge of the chisel is straight, one finds it difficult to keep the whole edge cutting, so that the corners take more than their share of the cutting action and thus wear round. If the edge is slightly convex as at (A), the middle of the cutting edge which has the support of the metal on each side of it takes the greatest strain.

Cast iron is a crystalline metal and when cut with a chisel breaks up against the inclined face of the chisel as it wedges its way into the metal to pry off the outer surface. Diagrams (1) and (2). The angle of the cutting edges for cutting cast iron should be from 65° to 70° and the inclination of the chisel will be approximately 35° so as to keep the lower face parallel with the surface being cut.

The keenness of the cutting edge of the chisel is important. It is advisable to stone it with an abrasive stone after grinding as a keen edge keeps sharp longer than a dull or ragged edge.

Method of chipping cast iron. After laying out the lines on the metal, the edge of the block shown in diagram (9) should be bevelled at about 45° all around down to the line to act as a guide and to prevent the edge from breaking off when chipping. A cape chisel should then be used to cut a series of grooves across the face, as drawn in diagrams (9) and (10).

The cape chisel, with its narrow cutting edge, concentrates the force of the blow and distributes it on a narrower cutting surface than the ordinary cold chisel, thus removing the hard skin with greater ease. The parts of the metal that are standing should be much less than the width of the cold chisel so that, when it is used, the corners of the chisel move freely along the two grooves made by the cape chisel.

The surface of the metal is finally finished with a sharp straight-edged cold chisel, cutting in various directions to reduce the eminences and produce a comparatively flat surface. The flat straight-edge of the finishing chisel forms a guide when cutting, because the flat face rests easily on the work and does not rock as it would if it were convex, as when roughing under a hard skin.

Chipping mild steel. Diagram (3). The great difference between chipping steel and cast iron is that the steel surface is removed in a curled chip because of its high tensile strength, but cast iron is brittle and crystalline and the chip breaks up. The angle of the cutting faces for mild steel would be from 30° to 35°. The mild steel is tough and requires a heavy blow to force the chisel into it. As in cutting mild steel on a lathe, the tool enters the metal and the chip is removed more easily if the end of the tool is lubricated by dipping it occasionally into oil.

The angle of inclination of the cold chisel is less for steel than for cast iron on account of the difference in the cutting angles. Compare diagrams (3) and (2).

Cutting a keyway. Diagrams (4) and (5). The chisel used for this purpose is a cape chisel with the lower facet longer than the upper facet to act as a guide in the keyway and to give more support behind the cutting edge than the ordinary cape chisel shown in diagram (9). An important feature of a cape chisel is that it is tapered back from the cutting edge (See plan Diagram 4) to provide clearance as it works in the groove being cut. In addition to this, the chisel is thick to give maximum sectional area of metal for a minimum width of cutting edge. The width of a cape chisel is determined by the width of the keyway or groove being cut.

Chisels with one facet are illustrated in diagrams (6), (7), (8), (11), (12), (13). These chisels tend to enter the metal in the direction of the face of the chisel nearest to the metal.

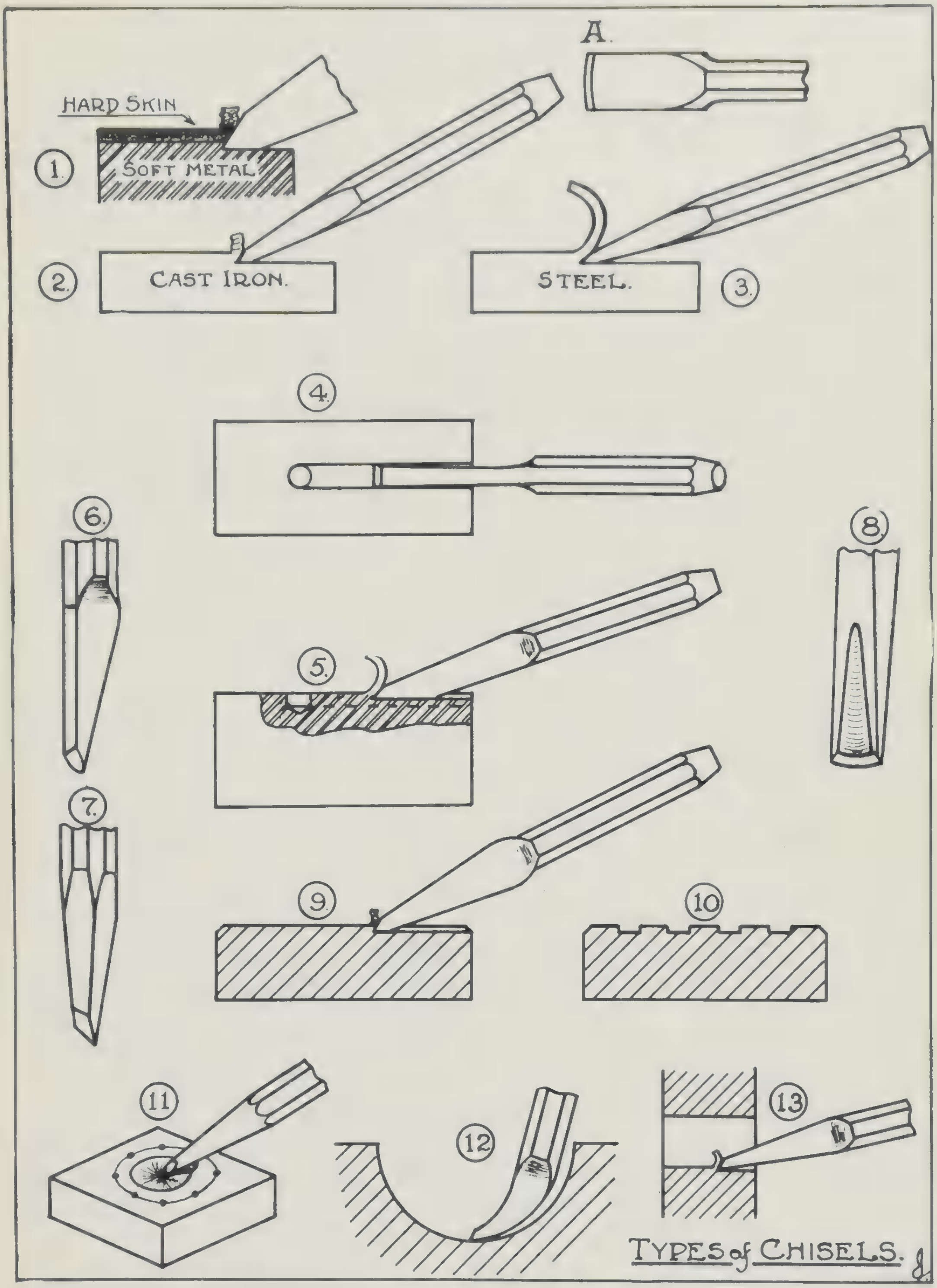
The gouge or round nose chisel. Diagram (6) is used for chipping round grooves, such as oil grooves in pulleys and bearings. A curved gouge is shown cutting an oil groove in a bearing, diagram (12). A small gouge or round-nosed chisel is illustrated in diagram (11) "drawing" a drilled hole, by cutting a groove on the side of the drilled countersink towards which the drill is required to drill.

The diamond point chisel. Diagram (7) is used to cut into right angled corners and is also used to correct errors in drilling holes, as shown in diagram (11).

The cow-mouth chisel. Diagram (8) is used for chipping and cutting circular work. It usually has a convex edge and a curved shape to suit the work being done.

Internal cutting with the chisel is illustrated in Diagram (13). Here the important thing is for the worker to see the cutting action. For this reason the cutting is accomplished while the chisel is only slightly inclined, which also gives the chisel an opportunity of cutting further into the hole. This one-sided chisel can also be used for squaring the bottom of squared holes.

General points in chipping. The eye of the operator should be upon the cutting edge of the chisel and not on the head of the chisel where the hammer strikes. Hold the hammer near the end of the handle and give it a free easy swing. Chip against the vise jaw, when possible, and



see that the work is supported by a block of wood or metal when held in the vise, so that it will not move down in the vise. While finishing, see that about 1/32″ is left for the cut, otherwise the chisel will not have sufficient metal to "bite" into, and will slip over the surface of the metal.

SOFT SOLDERING

Soft-soldering is not a difficult operation, provided that one is careful to observe strictly the considerations which are dealt with below.

Dirty or greasy metal. If an attempt were made to solder under these conditions, it would be discovered that the solder would gather in a lump and run over the surface of the metal, but would not unite with it.

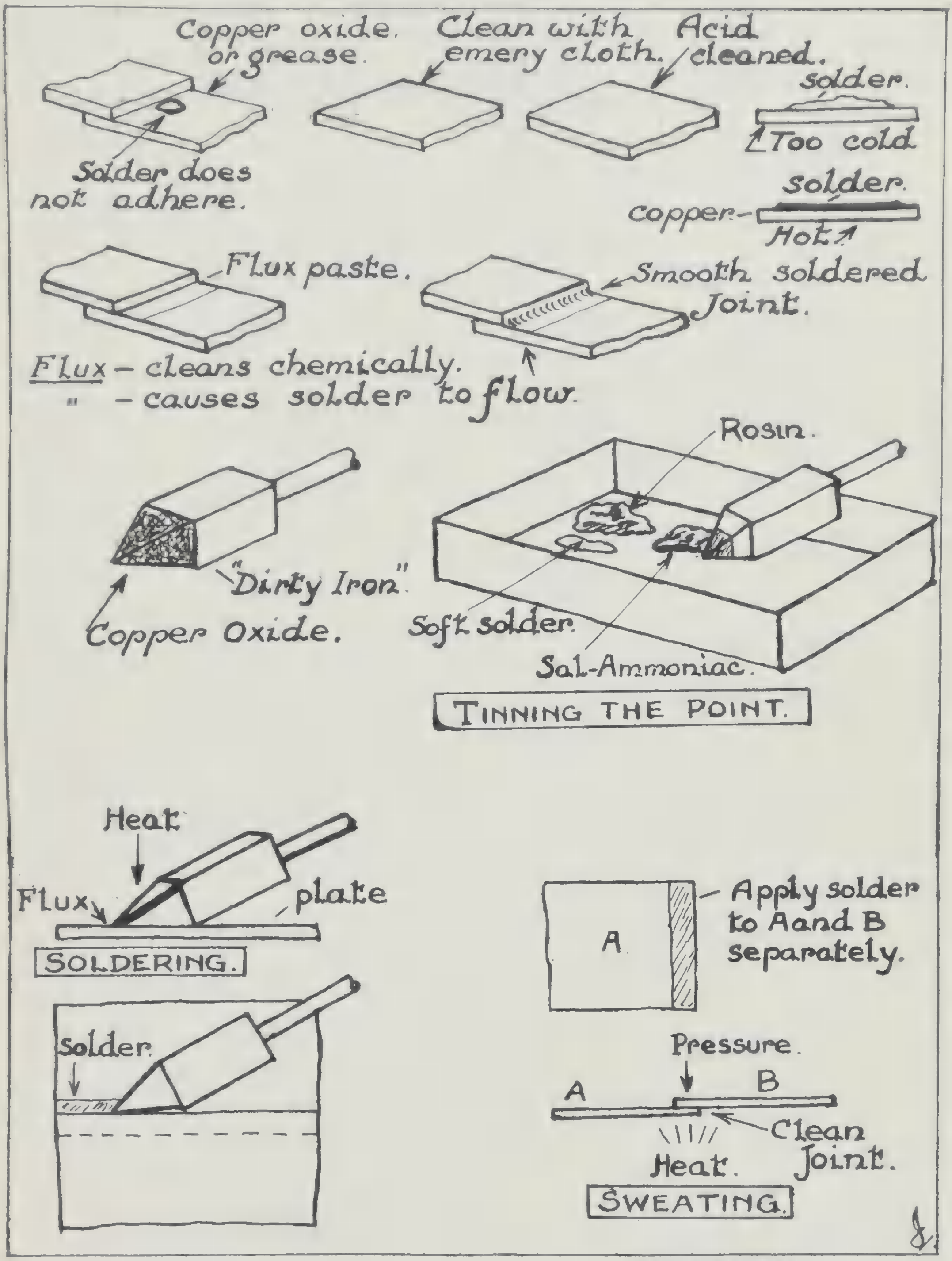
Cleaning the Metal. If the metal is covered with black scale (oxide), this can be removed with emery cloth. If copper, it can be cleaned by immersing the metal in a pickle; or by applying a little "killed spirits" (zinc chloride), making the metal chemically clean.

To produce a smooth, even, soldered joint. *First.* A flux must be applied such as "Fluxite" (a soldering paste), or "killed spirits", or rosin. Flux means "to flow", so that its functions are (a) to chemically clean the metal and (b) to cause the solder to flow. *Second.* If the metal being soldered is not hotter than the melting point of soft solder (half and half) which is 370° F, the solder will form in a quickly cooled pasty mass. This may be caused by passing a hot soldering copper too quickly over the joint. It may also be caused by the soldering copper not being sufficiently heated. It is very important, then, to use a hot soldering bit and allow it to rest on the joint until the joint is as hot as the bit, then it is moved *very slowly.* The solder will now remain molten on the hot joint for a few seconds and cool very slowly, producing a smooth, clean joint.

The soldering copper. A soldering copper should never become red hot, as it oxidizes very rapidly. Also, if it had been previously tinned, the tin would be burnt off and a "dirty iron" would result. The black scale so formed on the copper bit is called "copper oxide" and is a poor conductor of heat.

Tinning the point of the copper bit. First, the copper oxide must be removed by filing. When the bit has been heated so that it will quickly melt soft solder, it is tinned by rubbing it in a container holding Sal Ammoniac, Rosin and soft solder, or by rubbing it on a block of Sal Ammoniac in contact with a lump of soft solder.

Soldering. The position of the tinned copper bit is important when soldering. The face of the point of the bit should rest on the joint, as shown in the sketch, so as to obtain the conduction of as much heat as possible. This will pre-heat the joint and as the copper bit is moved slowly, the point leaves a smooth ribbon-like film of soft solder.


Copper oxide or grease.
Clean with emery cloth.
Acid cleaned.
solder.
Too cold
Solder does not adhere.
solder.
copper
Hot.
Flux paste.
Smooth soldered Joint.
Flux – cleans chemically.
" – causes solder to flow.
Rosin.
"Dirty Iron".
Copper Oxide.
Soft solder.
Sal-Ammoniac.
TINNING THE POINT.
Heat
Flux
plate
SOLDERING.
solder
Apply solder to A and B separately.
A
Pressure.
A
B
Clean Joint.
Heat.
SWEATING.

SOLDERING

Sweating. A sweated joint is a strong, clean joint. Solder is applied to each piece as shown at A and B of the sketch, then the pieces are pressed together after flux is applied, and the joint heated, which causes the solder to melt and combine, leaving very little solder exposed.

Soft Soldering—Metals and Fluxes Used

Metals	*Flux*	*Chemical Name*
Brass and copper	Rosin Sal ammoniac Zinc butter	Colophony Ammonium chloride Zinc chloride
Galvanized iron (zinc coated)	Raw acid	Hydrochloric acid
Iron and steel	Sal ammoniac	Ammonium chloride
Lead	Tallow Rosin	
Tin (Block)	Zinc butter	Zinc chloride
Tin plate (Iron coated with tin)	Cut muriatic acid Rosin Zinc butter	Hydrochloric acid cut with zinc Colophony Zinc chloride
Zinc	Raw acid	Hydrochloric acid

BRAZING AND SILVER SOLDERING

Brazing and Silver Soldering belong to a class of soldering known as "Hard Soldering" as distinct from "Soft Soldering". If the joint in metals requires to be strong and cannot be welded, hard soldering should be the method adopted to fasten the metals together. Most of the best brazing or hard soldering is done with silver solder.

Spelter is the metal used for what is commonly called brazing. This is the cheapest metal to use for forming the joint, but if silver solder is used with care it is not as costly as it at first appears. Brazing spelter is an alloy of copper and zinc, commonly called brass, and can be obtained in varying degrees of fineness in the grain form; but brazing wire is more easily applied as a general rule.

Silver solder is usually used for hard-soldering copper, brass and german silver, but can also be used on iron, steel, monel metal, bronze, nickel and stainless steel. Silver solder is an alloy and can be obtained in various mixtures of silver, copper and zinc according to the type of metal being soldered. The advantage in using silver solder is that it melts freely and flows easily when properly fluxed into the joint. It is much more fusible than brazing spelter.

Silver solders are strong, having a tensile strength varying from 40,000 to 60,000 pounds per square inch. The joints made of silver solder are often stronger than the metals which are joined.

Melting temperatures of silver solder range from 1325° F. to 1600° F., varying with the amount of silver contained. The higher the silver content, the lower the melting temperature and the greater the cost.

Silver-soldered joints are malleable, resist the effects of vibration and corrosion and have relatively high electrical conductivity; therefore they are suitable for electrical work.

General method for hard soldering. Diagram (1). It is essential that the parts to be joined are clean. This can be accomplished by the use of abrasive cloth or by pickling.

Use of the flux. The flux has a two-fold purpose, (a) to chemically clean the joint, and (b) to cause the metal to flow in the joint and prevent oxidation while this is going on. *Borax* is the flux used for hard-soldering with silver solder or spelter and is most conveniently applied in liquid form. Use only the best borax and dissolve it in boiling water, four ozs. to the pint. It can then be applied by means of a brush. It can also be used as a paste mixed with water, or fused borax may be used mixed with water or with alcohol and kept in a sealed container. For any metal the oxides of which are hard to remove, use a paste of boracic acid and borax with zinc chloride.

Heating the work. If the work is large, heat the outer mass away from the joint so that when the heat is applied to the joint it will not be dissipated. It is sometimes advisable to protect the work from draughts with an asbestos shield.

Heat the joint all over gently to prevent blowing off the flux and see that there is enough flux rather than too little. The flux lifts from the joint, Diagram (3). as it is heated, while the water is driven off, and it

is at this time that there is a danger of oxidation which will prevent the making of a good clean joint. When the borax melts at 1400° F.—it runs freely over the work and into the joint, making it chemically clean and ready to be soldered.

Applying the solder to the joint. The silver solder can be purchased in thin rolls, filed powder, coils of wire and strips. Spelter is usually used in grain or wire form.

Apply the solder to the work quickly, (Diagram 5) as soon as the borax melts. A tray, such as shown in diagram (7), can be used for a long joint or the grain or filings can be applied with a spatula. Diagram (9). The temperature of the joint should be steadily increased until the solder melts, (Diagram 6), when it flows very freely into the joint, following the flux. The spatula can be used along the joint but it is not often necessary. If used, the spatula should be heated and coated with the solder.

As soon as the joint is completed and has cooled, remove the work from the heat, and plunge it into cold water. This removes the flux and scale. If left to cool outright, the borax vitrifies and is difficult to remove. A small quantity of sulphuric acid added to the water used for quenching assists in the removal of the scale and flux.

The joints for hard soldering should be tight, as the solder, when melted, is very fluid and runs into the finest space. If the joint is not tight, the solder runs through and is wasted.

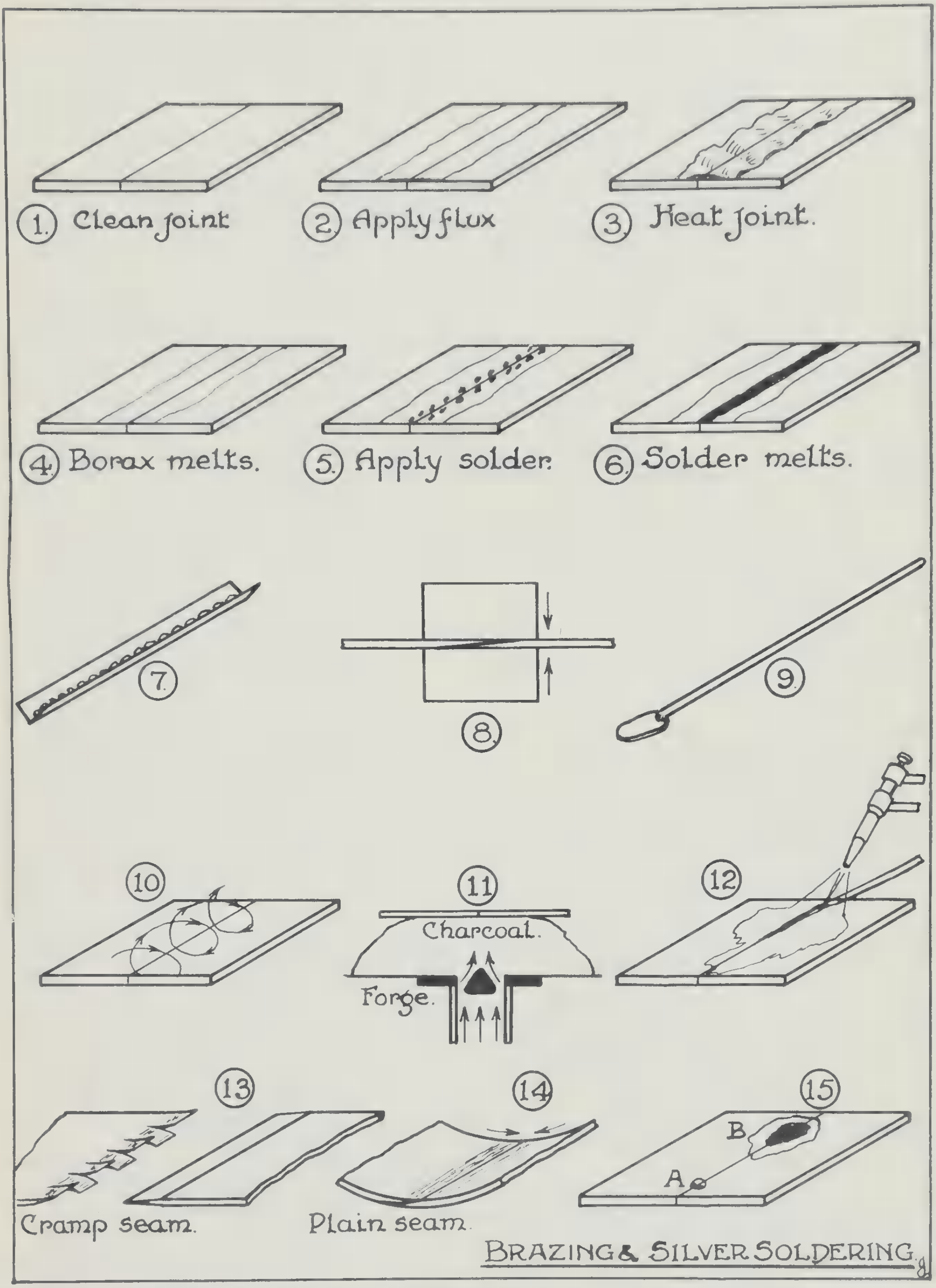
Methods of heating the work to be joined. The silver soldering of small work is usually performed with a blowpipe, as shown in diagram (12). The joint should be pre-heated carefully by moving the flame around the joint, as shown in diagram (10). Do not impinge the flame too much on the joint or the flux will be blown off.

For brazing, an oxy-acetylene torch may be used, if available. Brazing can also be accomplished by the use of a good gas blow torch. A good charcoal forge fire with an air blast provides a clean method of heating a joint for brazing, as shown in diagram (11). Be sure that the fire is deep, as shown in diagram (11), so that the air will not come in contact with the work and oxidize it. A band saw can be hard soldered by the application of red hot tongs, which heat and hold the joint together. (Diagram 8).

Types of joints. For ordinary work, a good butt joint is satisfactory, but if the work is to be strained very much a seamed joint is necessary.

Diagram (13) shows a "cramp seam". Here the two sides of the joint are tapered off and one side is cut. Then the alternate parts are pushed up and down to receive the other beveled side of the joint. The joint is now closed ready for hard soldering. The cramp seam is a very strong joint. The next joint in order of strength is the plain seam, diagram (14), which is beveled on both sides of the joint by filing one face on each side. It is necessary in both the cramp and plain seam joints to wire the job with binding wire to prevent it from opening when hard soldering.

Balling, or Bunching, is the melting of the silver solder or spelter into a ball form which runs on the work but will not adhere to it. It is



(Diagram 15. A shows "balling", B correct flow of solder.)

a common occurrence with the beginner in hard soldering, and usually indicates improper fluxing or insufficient heat on a dirty oxidised joint. To correct it, the work should be removed from the heat, properly cleaned, re-fluxed and then sufficient heat should be applied.

SCRAPING

Scraping is essentially a precision operation to remove very small quantities of metal, in order to produce flatness or to obtain surfaces that give maximum contact with each other. If the surface of a machined flat block of metal is examined before and after scraping, it will be seen that, before scraping, the metal surface is full of eminences and depressions, but after scraping the serrated and matted surface is reduced to a smooth fine surface. After scraping, the molecules in two metal blocks, placed together, are very close to each other to give to some extent molecular attraction. This is very noticeable in standard gage blocks which adhere to each other with an adhesion that resists 2½ times atmospheric pressure. It is impossible to attain such a standard of flatness and closeness of the metal surfaces by hand scraping alone; but this should be kept before the beginner as an objective.

Preparation of work before scraping. If cast iron work is intended to be scraped, it must be understood that cast iron warps considerably after the hard skin has been removed by machinery, because the removal of the hard skin allows the internal strains to work out of the metal.

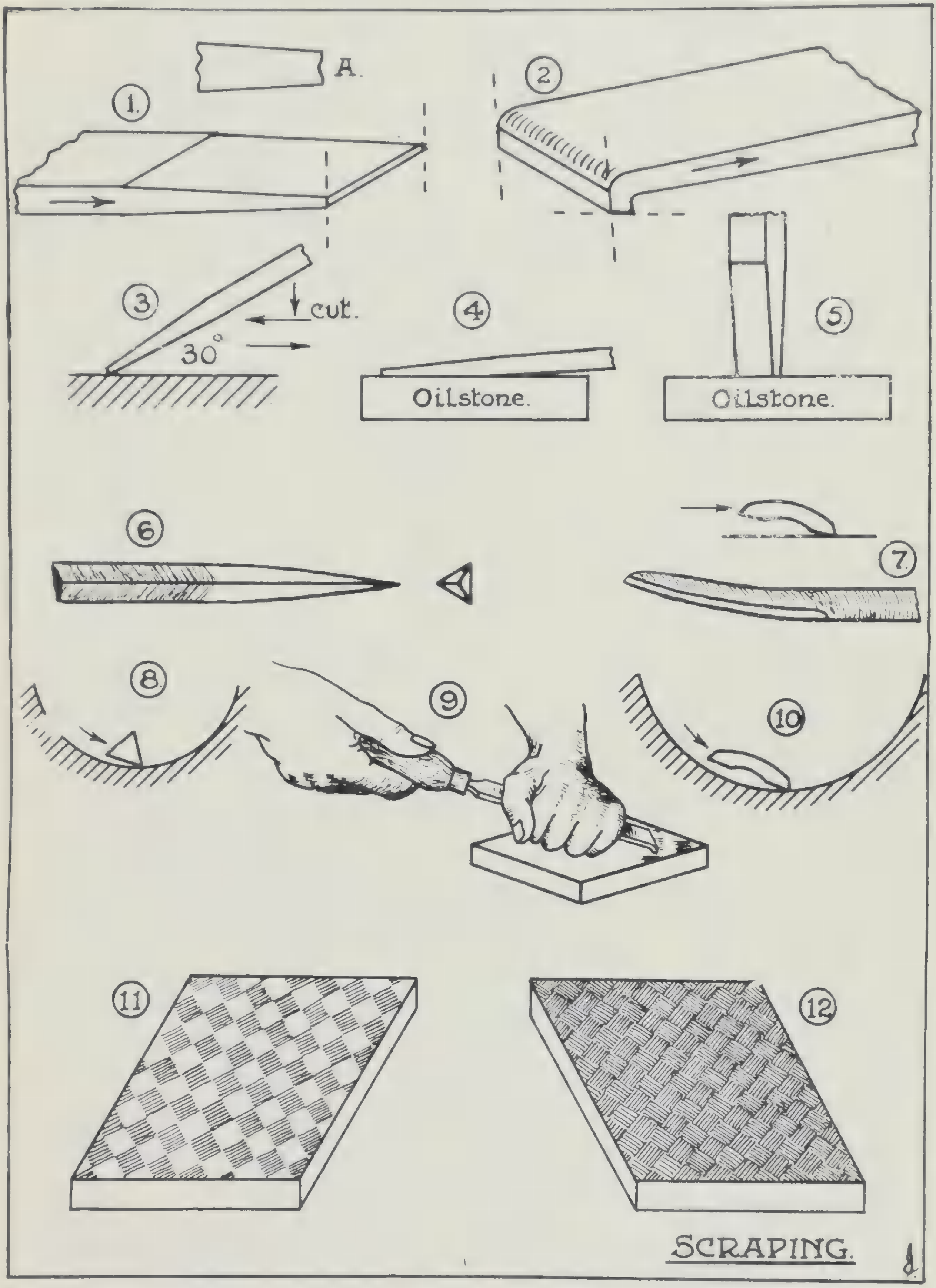
It is advisable to remove the skin of the metal, then allow the metal time to season, then to take a final finish cut before scraping.

It is advisable to remove with a file all feed serrations left by machinery before scraping is started, as the scraping would take too long to do this.

The flat scraper. Diagram (1) can be made out of an old 8″ or 10″ flat file, but special scraper steel can be purchased from which the scraper can be made. If a file is used for the scraper it must first have the teeth ground off on all sides back about 3″ from the end, and then be heated carefully and forged at a heat suitable for high carbon tool steel. After grinding the taper to an edge about 1/32″ to 1/16″ thick, it should be hardened carefully and the strains taken out of it by a slight temper heat so that a drop of water will just bubble on the surface of the metal. The cutting edges of the scraper are now carefully ground exactly straight and square or slightly concave, as shown at (A). It is necessary now to oilstone the flat surface and the edge surface to produce two sharp, keen cutting edges.

The scraper is rubbed flat on the oilstone, as shown in diagram (4). Then it is held vertical to the oilstone surface and the blade turned to a position of 45° to the direction of the motion of the stoning. This position of the scraper tends to produce an edge that is slightly convex lengthwise of the edge, but square across the edge. The scraper that is slightly convex picks out the eminences on the work without the corners cutting into the surface of the work and is an advantage in rough scraping; but a dead straight scraping edge should be used for the final finish operation.

Scraping the work. Diagram (3) shows the flat block. The angle of the tool is approximately 30°, but the angle must be determined by "feel" in actual use. If the tool is inclined too much, the scraper edge


A.
1.
2.
3.
cut.
30°
4.
Oilstone.
5.
Oilstone.
6.
7.
8.
9.
10.
11.
12.
SCRAPING.

will "bite" into the work. Incline the tool so that the edge will cut a small particle easily without slip. Short strokes are taken less than ½″ in length. A slight pressure is used on the forward or cutting stroke which is relieved on the return.

Dip the scraper occasionally in turpentine and keep greasy fingers from touching the work plate.

A true plane surface plate is used to check the work. This precision tool should be kept protected by its wooden cover when not in use. The surface of the plate should be rubbed over evenly and thinly with red lead, prussian blue, or drop black. The work to be scraped should now be rubbed on the surface plate and the high spots will be seen on the work. Rub the work all over the surface of the surface plate equally with a circular motion. The marked spots should then be scraped off and rubbed again on the surface plate for a second test. As the plate improves in flatness a very thin application of the marking material should be used.

The work should be scraped in approximate squares, as shown in diagram (11) ; then the spaces not scraped by the first operation should be scraped, completing the surface by scraping in opposite direction, as shown in diagram (12).

The hook scraper, shown in Diagram (2), is sometimes used to give the "flower" finish to work being scraped. Light pressure should be used when scraping in the direction shown in diagram (2). This type of scraper is also used for bearing surfaces where it would be impossible to use the straight flat scraper.

The method of holding the scraper, is illustrated in Diagram (9). The tool is held firmly to keep it under perfect control to carry out such a precise operation. The upper part of both arms should be supported by the body and the forearm muscles should be mostly used to control the tool.

The three-cornered scraper, Diagram (6), is often used to remove the sharp edge of a curved surface or hole. It is usually made from an old file slightly convex lengthwise but flat across its surface. It is oilstoned to produce a sharp keen edge. If used on curved work, such as illustrated in the bearing diagram (8), it is advisable to bend the tool to a curve, as in diagram (7), to allow the operator to work conveniently into the bearing.

The half round scraper, Diagram (7), is made from a half round file bent to a convenient shape, or hollow ground on the under side, as shown, to make the stoning of the edge easier.

Scraping bearings. Diagram (10). The half round scraper is usually used in the direction shown. Sometimes a hook scraper, diagram (2), is used. The edge is formed to fit the curvature of the bearing and then sharpened to cut on the pull or draw stroke.

BENCHWORK QUESTIONS

1. Why are bearings babbitted?
2. What does Babbitt metal commonly consist of?
3. What is the special reason for the use of each metal that is used to make up Babbitt metal?
4. How is Babbitt metal prevented from moving in the bearing?
5. Sketch a section of a solid type of bearing as set up for babbitting.
6. Illustrate by sketches how a split bearing is set up for babbitting.
7. What happens if the bearing is not pre-heated before the babbitt is poured?
8. How can the molten babbitt be prevented from unduly oxidising?
9. Why is it necessary to be certain that sufficient babbitt is heated to completely pour the bearing?
10. How would you test to see that the babbitt is the proper heat for pouring?
11. How are the oil grooves put in the babbitt?
12. Make a sketch of a single and double belt on a pulley showing the direction of the scarfs and the rotation of the pulley.
13. Why is the grain side of a belt run next to the pulley?
14. Sketch two opposite sides of a belt showing a leather-laced joint.
15. What is the best type of belt joint?
16. State the advantages and disadvantages of at least two types of metal belt joints.
17. Why is it necessary to cut a belt joint absolutely square?
18. How would you measure the length of belt to be cut to drive a machine from a countershaft?
19. Sketch a cold chisel cutting Cast Iron and one cutting machine steel; show the cutting angles of each chisel.
20. State the differences between a chisel ground at a cutting angle of 50° and one ground at 70°. Indicate by arrows the direction of forces tending to separate the metal.
21. Why is a cape chisel made wider at the cutting edge than the body of the metal behind the cutting edge?
22. Sketch 4 distinct types of chisels stating their uses.
23. Why is copper used in making a soldering iron?
24. Name the fluxes used in soldering iron, galvanized iron, copper, and tin plate.
25. What are the functions of a flux?
26. Describe the "tinning" of a soldering iron.

27. Of what does soft solder consist?
28. What are the causes of solder not adhering to the metal being soldered?
29. Describe with sketches the operation of sweating two pieces of metal together. What are the advantages of this process?
30. What is the black scale called that forms on a soldering iron? How can its formation be prevented?
31. What are the advantages of: (a) Soft Soldering? (b) Hard Soldering?
32. What is the chief flux used in hard soldering? How is this flux prepared and used?
33. State clearly, with the aid of sketches, definite steps in the process of hard soldering.
34. Sketch two forms of scrapers used for scraping a cast iron surface plate.
35. How is a scraper sharpened? Why is the end of a flat scraper made narrow in thickness?
36. How is a block being scraped tested for flatness?
37. Show by sketches the method of scraping a flat block.
38. Sketch two forms of scrapers used for scraping babbitt bearings.
39. Sketch a flat scraper in elevation showing its angular position in relation to the surface of the work being scraped.

DRILLING

THE PARTS OF A TWIST DRILL

Twist drills are made by milling two spiral grooves, called the "flutes", in a piece of carbon tool steel or high speed steel. Occasionally they are made by twisting a flat piece of steel; hence the expression "twist drill". The common shanks used to drive drills are, (1) The bit stock shank; (2) The straight shank; (3) The taper shank; (4) The ratchet shank. The straight shank is used for small drills when driven by being held in a drill chuck, but larger drills which have heavier work to do are provided with a taper shank.

The taper shank wedges itself into the drill spindle or socket and lines itself axially true with the spindle. In addition to this the increased pressure on the drill point tends to wedge it tighter into the spindle. This friction drive provided by the taper is insufficient to prevent the drill from slipping under a heavy cutting load, so that a positive drive is obtained by fitting the flat tang into a corresponding slot in the spindle.

The shank. Diagram (4), is made with a "Morse Taper", the number or size of the taper depending on the size of the drill. Example No. 1 Morse taper for small drills and No. 2 for larger drills, etc

The size of the drill is usually stamped in the recess between the shank and the body of the drill and high speed drills are usually so marked.

The body of the drill is usually made slightly smaller near to the tang, to give the drill a very slight clearance back from the point.

The lands of the drill are given a clearance to prevent rubbing and heating up the drill. A narrow strip called the "margin" is left concentric to guide the drill and ensure that the hole drilled is accurate as to size.

The flutes, or spiral grooves, are cut in the drill to provide the cutting edges at the point and are so shaped that the chip removed is curled, so that it will occupy a small space and come out of the hole. The flutes provide a means of lubricating the drill when cutting.

The web of the drill, Diagram (1), is the strip of metal which separates the flutes and is thicker towards the shank; hence, worn drills require more thinning than new drills. Diagrams (2) and (3) show the plan and elevation of a drill with the relation of the cutting

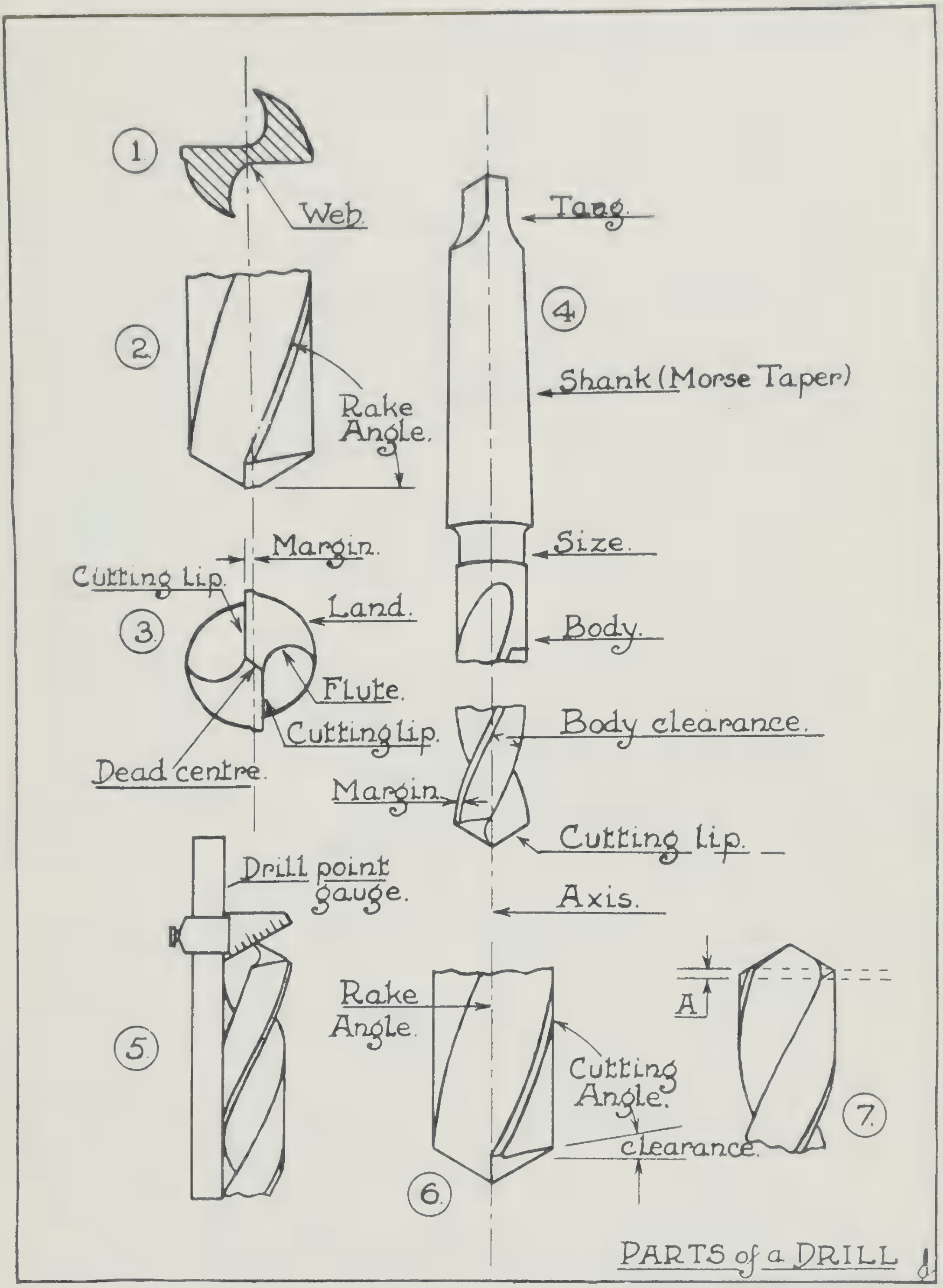
edges and the end of the web at the point which is called the "dead centre". This *dead centre* is the most troublesome part of a drill to grind.

Testing the drill point. Diagram (5) shows one type of drill point gauge. The angle of the cutting edge should be at 59° inclination to the axis of the drill. The gauge shown checks this angle and also measures the length of the cutting edges to determine if they are equal.

Diagram (6) shows the similarity between a drill and a lathe tool. The cutting angle is governed by the angle provided by the spiral flute and the clearance angle back of the cutting edge. The rake angle is governed by the inclination of the spiral flute to the axis. A straight flute drill with no rake angle is generally used for brass, as a lathe tool has no rake angle for turning brass.

The cutting angle of the drill is reduced in drilling soft metals by increasing the clearance angle.

The amount of clearance of a drill point can be observed by comparing the variation between the height of the back corner of a flute and the outer corner of the cutting edge in the same flute as shown at A in diagram (7).


1.
Web.
Tang.
4.
2.
Shank (Morse Taper)
Rake
Angle.
Margin.
Cutting lip.
Size.
Land.
3.
Body.
Flute.
Body clearance.
Cutting lip.
Dead centre.
Margin.
Cutting lip.
Drill point
gauge.
Axis.
A
Rake
Angle.
5.
Cutting
Angle.
7.
clearance.
6.
PARTS of a DRILL

DRILL GRINDING VARIATIONS

Drill grinding, like all other tool grinding, requires a distinct knowledge on the part of the grinding operator if the drill is to do efficient work. It is quite common to see drills in use which are not drilling round holes or holes true to the nominal size of the drill. Machine grinding of drills larger than 3/8″ is more efficient and economical, while drills 3/8″ and smaller are usually ground by hand.

The point of the drill, diagram (1), is usually made with the cutting edges at an angle of 118° to each other. The point of the drill is defined as the entire cone-shaped surface at the cutting end of the drill.

The clearance of the lips or cutting edge is most important and varies from 12° to 15°, as shown in diagram (1). Lip clearance is the relief which is given behind the cutting edges to allow the cutting edges to enter the metal when cutting.

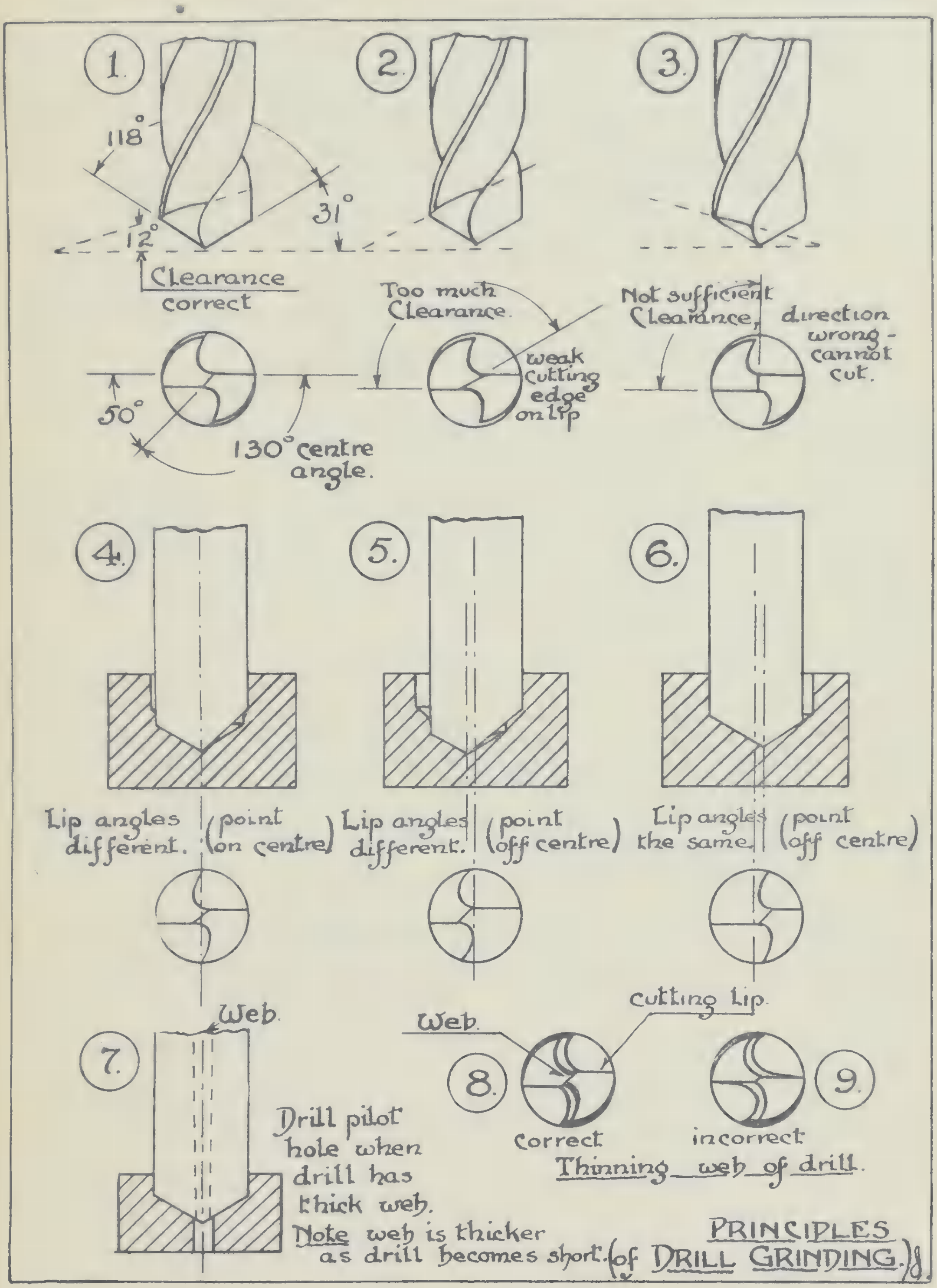
Diagram (2) shows a drill with too much lip clearance or relief. The effect of this will be to weaken the cutting edges so that they heat up quickly when cutting.

Diagram (3) shows a drill with no clearance back of the cutting edge, but rather the opposite, so that it would be impossible for the drill to cut, because the drill point is rubbing on the two surfaces behind the cutting edges.

To detect correct clearance angles. The drill point can be observed for clearance, as in diagram (1), but in addition to this one can examine the angle of the line across the dead centre of the drill. The edge should indicate not less than 120 degrees and not more than 135 dgrees. The angle of the lip clearance should be gradually increased as the centre of the drill is approached. This feature will be explained fully in the next lesson.

Angles of lips in relation to axis of the drill. Diagram (4) shows the two lips of a drill ground at different angles to the axis of the drill. In consequence of this one edge only can cut, which means that the drill will be cutting out of balance and will probably drill a hole that is neither round nor of the correct size.

Diagram (5) shows a drill with cutting edges ground at different angles with this axis of the drill. In addition to this, the dead centre at the extreme tip of the end of the drill is offset from the axis of the drill. The result of these two defects will be that the drill will cut "unbalanced", which will produce: (1) A hole out of round, (2) A hole


1.
2.
3.
118°
31°
12°
Clearance correct
Too much Clearance.
Not sufficient Clearance, direction wrong - cannot cut.
weak cutting edge on lip
50°
130° centre angle.
4.
5.
6.
Lip angles different. (point on centre)
Lip angles different. (point off centre)
Lip angles the same. (point off centre)
Web.
7.
Drill pilot hole when drill has thick web.
Note web is thicker as drill becomes short.
Web.
cutting lip.
8.
9.
correct
incorrect
Thinning web of drill.
PRINCIPLES of DRILL GRINDING.

much larger than the drill, (3) A hole that is drilled offset from the original location.

Diagram (6) shows a very common error in drill grinding. The cutting edges are correctly ground to the angle with the axis of the drill, but one cutting edge is longer than the other and the dead centre is offset from the axis. Some mechanics do this purposely to drill holes larger than the nominal size of the drill and to make it work more easily. The chip pressure on the cutting edges is varied, therefore the drill is out of balance and tends to produce holes out of round and oversize.

Pilot holes, diagram (7), are small holes drilled before using a large drill, because the web of any drill does not cut quickly and usually retards the cutting action of the edges of the lips. If a small pilot hole, slightly larger than the thickness of the web, is first drilled, the drill will cut quickly and retain its true location.

The web of a drill, as shown by dotted lines in diagram (7), is the part of the drill which separates the flutes and increases in thickness towards the shank to give the greatest strength to the drill where the greatest strain would occur.

Thinning the point. Drills which have the extreme point of the drill thinned will feed more easily into the metal being drilled. This operation requires care and the use of a round-faced abrasive wheel. The grinding should be in the curved part of the flute, as shown in diagram (8) and not running into the straight part of the flute, as shown in diagram (9), as this will cut across the rake angle of the spiral flute of the drill. Do not weaken the drill by grinding too far up the flutes.

Summary: 1. Insufficient clearance causes drills to split.

2. For soft metals the drill should have more clearance than for hard metals.

3. Keep the cutting lips of equal length and of equal angle with axis.

4. Keep cutting edges as nearly straight as possible.

5. The clearance angle of a drill point varies, increasing from the outside towards the point.

THE THEORY OF DRILL GRINDING

Most drill operators seem to understand readily how a drill should be ground, but experience a great difficulty in carrying out the ideas that are in their minds. Where a special drill grinder is available, most mechanics have little trouble in grinding a drill successfully, but where hand grinding has to be carried out difficulties often arise. The main difficulty seems to be in knowing just how to move the drill in relation to the flat face of the cup wheel.

It is the intention of this lesson to attempt to explain just what the drill movement must be.

Diagrams (1), (2) and (3) illustrate three different movements which may be given to a drill and show the effect of such movements on the drill point.

Diagram (1) shows a drill moved in such a manner that the surface of the point ground would form a part of the surface of a cylinder. This would give a similar clearance from every point back of the cutting edge and, as will be seen later, this would be incorrect.

Diagram (2) shows a drill point being formed the surface of which is enveloped by a truncated cone. In consequence of this, there will be a varying clearance, increasing as it approaches the dead centre. This is the correct condition.

Diagram (3) shows a drill point being formed the surface of which is enveloped by an inverted truncated cone, which gives greater clearance at the outside than at the point. This condition is decidedly incorrect and is the reverse of the desired condition as shown in diagram (2).

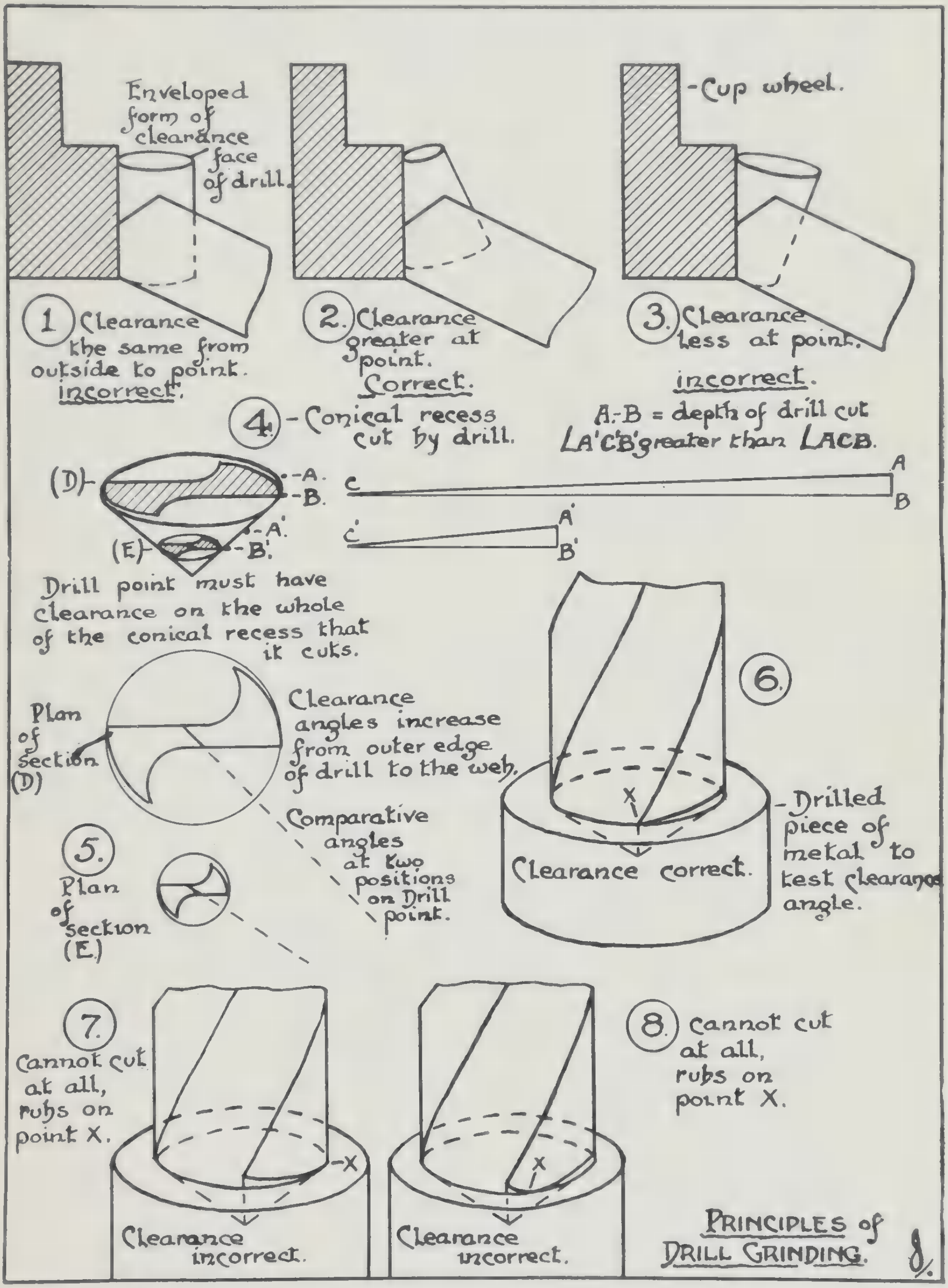
The working conditions of a drill point are illustrated in Diagram (4). A drill point runs in a conic recess, which it produces by its cutting action. It is therefore necessary that every part of the drill point behind the cutting edges have sufficient clearance to prevent rubbing the conic recess, which would keep the cutting edges from doing their work effectively. Two horizontal sections of a drill point are shown in diagram (4). If the upper section moves down a distance A. to B., in cutting one revolution, then the lower section must move a similar distance A'B'. A diagram shows a circumferential stretchout of the progression, from A to B and from A' to B'. It will be observed that angle A' C' B' is greater than angle A. C. B., which proves that the clearance must increase as it approaches the dead centre, because of the condition in the conic recess and the progression of the drill. The increase of the angle of clearance is shown by the dotted lines in diagram

(5). Although the angle of clearance at the outside of a drill may be 12° to 15° it will increase constantly as it approaches the dead centre, otherwise the clearance would not be sufficient to maintain equal cutting conditions in the conic recess.

Test for clearance. If a conic recess is drilled in a piece of metal, as shown in diagram (6), (7) and (8), and a little prussian blue is rubbed over the surface, it may be used as a test piece. The drill is held vertical and rotated backwards. When removed, it can readily be seen which part of the drill point is in contact. The blue should only show on the two cutting edges, as at X in diagram (6).

If the blue shows at X in diagram (7), then there is no clearance at all. If X in diagram (8) shows blue, it shows that there may be a general clearance behind the point, but the cutting edge would not be allowed to cut.

Conclusions. From the foregoing one can see that a drill when being ground must be presented to the stone to give the proper angle at the point, but by far the most important thing is to so turn and swing the drill that an increasing clearance may be obtained on the drill point behind the cutting edges.


Enveloped form of clearance face of drill.
-Cup wheel.
1 Clearance the same from outside to point. incorrect.
2. Clearance greater at point. Correct.
3. Clearance less at point. incorrect.
4. - Conical recess cut by drill.
A-B = depth of drill cut
∠A'C'B' greater than ∠ACB.
(D)
-A.
-B.
-A'.
(E)
-B'.
C
A
B
C'
A'
B'
Drill point must have clearance on the whole of the conical recess that it cuts.
Plan of section (D)
Clearance angles increase from outer edge of drill to the web.
Comparative angles at two positions on Drill point.
5. Plan of section (E)
6.
X
Clearance correct.
-Drilled piece of metal to test clearance angle.
7. Cannot cut at all, rubs on point X.
-X
Clearance incorrect.
X
Clearance incorrect.
8. cannot cut at all, rubs on point X.
PRINCIPLES of DRILL GRINDING.

DRILLING QUESTIONS

1. Sketch a taper shank drill showing the following parts:—Tang, taper shank, body, body clearance, margin, land.
2. Sketch the end view of a drill point showing:—Dead centre, flute, cutting lips, land, margin.
3. Sketch a twist drill showing the rake angle, cutting angle and clearance angle.
4. Why is a brass drill provided with a straight flute?
5. How is a drill point tested for accuracy when grinding?
6. What is the correct angle for a drill point?
7. Sketch an end view of a drill point showing the correct angle of clearance.
8. Make sectional sketches of a piece of metal being drilled by a drill with:—(a) Cutting edges at different angles to the axis of the drill. (b) One cutting edge longer than the other but correct angles to the axis.
9. Make a sectional sketch of a piece of metal being drilled after drilling a pilot hole.
10. Sketch an end view of a drill showing the thinning of the point.
11. If a piece of paper were wrapped against the ground face of a drill point what form would the paper take?
12. What is the difference, if any, between the clearance angle near the outer edge of a drill and the clearance near the centre?
13. What is the most inefficient part of a drill point when cutting?
14. If the outer edges of a drill are rounded when drilling, what does it indicate?
15. Does a drill make a perfectly round hole?
16. Does a drill usually drill a hole accurate as to size?
17. How can a drill be made to drill oversize if necessary?

LATHEWORK

THE FEED MECHANISM OF THE LATHE

It is very necessary that one should understand thoroughly the feed mechanism of a lathe, in order to be able to use the lathe intelligently. One should be able to visualize the workings beneath the apron at all times, for different uses. The apron mechanism illustrated on the opposite page is a very simple type and quite commonly used in low-priced lathes. One should remember, of course, that there are many different types of lathe apron mechanisms, and when the one illustrated is properly understood, one should seek out others and investigate their workings.

Apron mechanism set for cross feed. Diagrams (1) and (2). The view shown at diagram (1) is seen from the back of the apron looking towards the front of the machine and only the skeleton mechanism is illustrated. The entire apron frame is removed in diagram (2) to simplify the diagram. Rotation is given to the lead screw (which is splined to function as a feed rod as well) by connecting it through gears to the lathe spindle (see page 47). The relative speed of this rotation is governed by the gear ratio and may be changed as required. *A worm* (held between two brackets) is always in mesh with the splined shaft and is driven around by a sliding key which allows the worm to move along the shaft freely with the apron.

A worm gear is always in mesh with the worm and therefore always rotates when the spline shaft rotates. The back of the worm gear is conically recessed and receives a cone clutch, which can be engaged or disengaged by turning the clutch knob at the front of the apron.

When the clutch knob is tightened, a shaft is fastened to the clutch and drives the pinion A diagram (2), which is always in mesh with the idler gear B. This idler gear may be rocked around the centre of the shaft pinion A by lifting the lever shown in diagram (1) to change from cross to long feed or into neutral position.

For cross feed the idler gear B is connected to a pinion (C) which is fastened to a gear (D) which is in mesh at all times with a pinion (E) which is fastened to the cross feed screw which moves the cross slide. The direction of the cross slide movement inwards or outwards may be changed by changing the reverse gears on the end of the lathe. When using the automatic cross feed, the carriage should be clamped to the lathe bed to assure the work being faced square.

Apron mechanism set for long feed. Diagrams (3) and (4). The lever arm that carries the idler gear (B) is now moved up as shown in diagram (3). This movement disconnects the idler gear (B) as set for cross feed and connects it with gear (C), which has fastened to it a pinion (D) which is always connected to the rack which is fastened beneath the ways of the lathe bed. As the pinion (D) turns, the carriage is bound to move on the ways of the lathe bed, giving automatic long feed. Diagram (4) shows the side view of the gears and the pinion meshing with the rack.

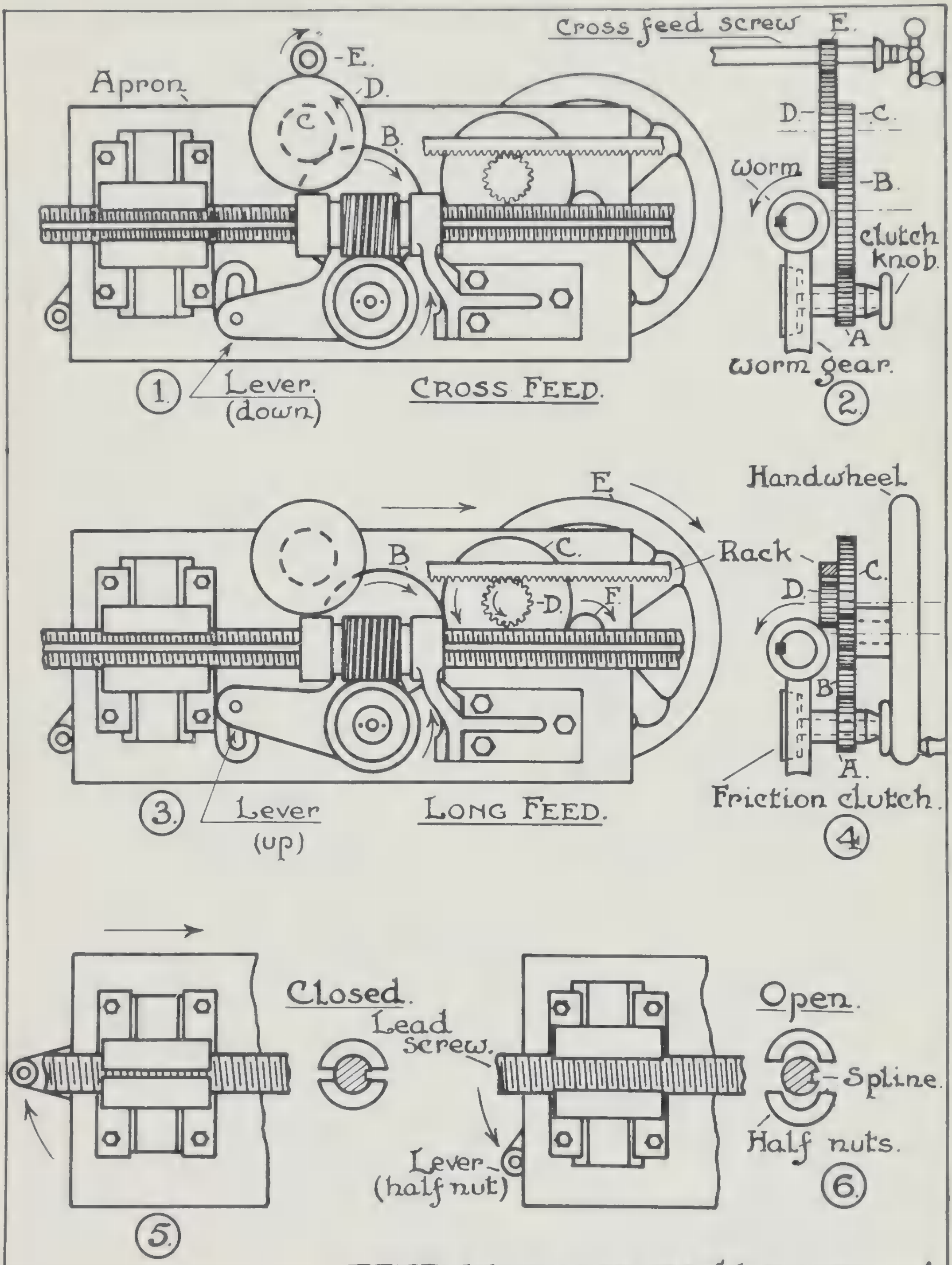
Hand long feed. Diagram (3). If the handwheel E is turned, the pinion (F) which is fastened to the handwheel meshes with the gear (C), which moves the pinion (D), which is in mesh with the rack and causes the carriage to move along the bed.

Use of the half nuts. Diagrams (5) and (6). The half nuts give a positive feed to the carriage and should only be used for screwcutting on the lathe. When the lever is lifted up, as shown in diagram (5), the half nuts are engaged with the threads of the lead screw and the carriage is forced to move as the lead screw rotates.

Diagram (6) shows the half nut lever down and the half nuts disconnected from the threads of the lead screw. On most lathes, it is impossible to connect the half nuts while the idler gear is in mesh with the gears for automatic long or cross feeds.

It is very important that the friction feed be disconnected while the half nuts are engaged in order to prevent damage to the machine on account of the difference in the rate of the movement of the carriage in the two different set ups.

The friction feed cannot be used for screw cutting because it is driven through a friction clutch and, therefore, is not regular and positive so as to guarantee threads of an even regular lead.



FEED MECHANISM of LATHE.

FORMING

This lesson deals with forming a piece of work in an ordinary lathe when one or two pieces are required.

Special forming tools, such as are used on turret lathes, are referred to on page 129.

Owing to the definite directions given to the ordinary lathe tool by the long, cross and compound feeds, forming an irregular profile on a piece of work offers the operator something at least different, if not difficult.

The work taken to illustrate a typical forming job on centres, is in the form of an ordinary machine handle. After turning down to the outside diameters of the form, as shown in diagram (1), a form tool can be ground from an ordinary tool bit to shape the part as shown. The other part of the concave form can be cut with an ordinary round-nosed tool.

Method of forming. It is better to rough form to shape in a series of steps, as shown in diagram (2), using the automatic long feed and visualizing the future outline of the job with the drawing in front of the operator.

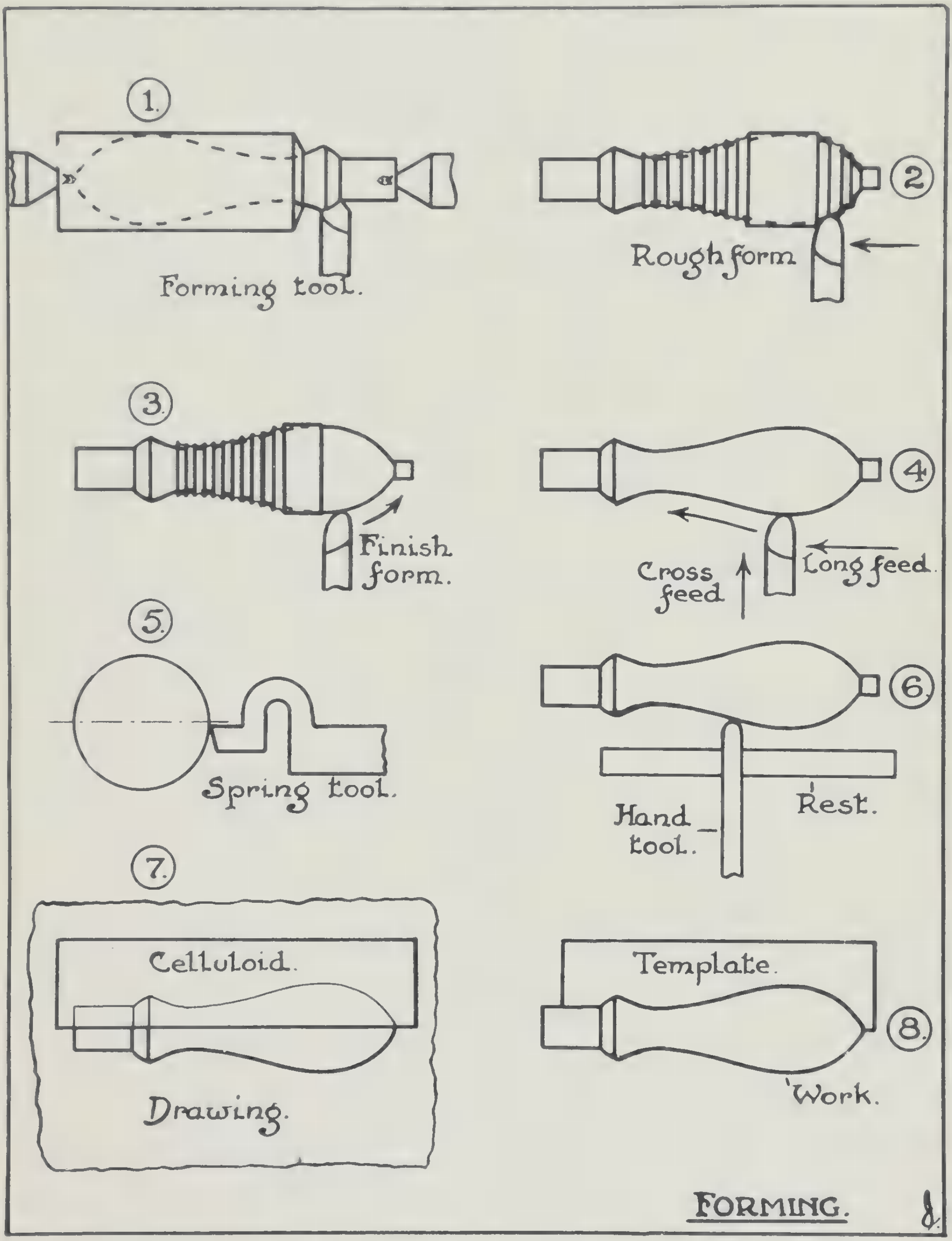
To finish form. It is advisable for the operator to work down the form and not up, as there is a danger when working up, that the tool may "run in" to the form, but when working down, it can only "run out" and thus keep the form oversize.

Free forming by hand. Diagram (3). By this method the operator controls both long and cross feeds and it may take some time before sufficient skill and combination or co-ordination of long and cross movements are developed to produce a harmonious curve.

Forming with automatic long feed. Diagram (4). This method is simpler than the previous one and the operator is advised to try this method first.

Place a piece of paper beneath the work so that the form shows up clearly against the white background. Put in the fine automatic long feed and start at the greatest diameter and feed the tool inwards, down the form, by a hand cross feed.

To produce a smooth finish. A spring tool may be used as shown in diagram (5) and the irregularities may be smoothed down by such a tool without digging into the work. To produce free curves, the old-fashioned hand tool, as shown in diagram (6), may be used to advantage,


1.
Forming tool.
2.
Rough form
3.
Finish form.
4.
Cross feed
Long feed.
5.
Spring tool.
6.
Hand tool.
Rest.
7.
Celluloid.
Drawing.
Template.
8.
Work.
FORMING.
J.

as it gives the operator the necessary freedom to follow the varying changes of the form.

To polish the work. Use abrasive cloth in strips pressed against the work held in tension with the right and left hand. The cloth fits itself to the form, producing a smooth finish, or the cloth may be pressed against the work by the fleshy part of the hand, which cushions itself to the form.

Making a template from a drawing. A very effective and simple method of doing this is to take a piece of thin celluloid, as shown in diagram (7), and place one straight edge on the centre line of the drawing. Scribe the outline in the celluloid with a scriber, then place it on a smooth block of wood and cut part way through the celluloid with a sharp well pointed knife. The celluloid can now be snapped in two with the fingers and the formed outline smoothed with a fine file or emery cloth. (Scissors may sometimes be used.)

Testing the work for true form. Diagram (8). The celluloid pattern can be placed over the work and the form checked approximately by looking through it, or the work can be checked accurately by a fit test. It is much better, however, to use a thin metal template to check the work if many are to be made, or if the work must be very accurate.

THE MANDREL

A mandrel or arbor is used to mount work on, so that it may be driven between centres.

A hole is usually drilled and reamed in the work and the mandrel is forced into it. A mandrel must be accurately made to produce accurate work turned on it. If the mandrel surface were not perfectly concentric about its axis, the work mounted and produced on it would not be true about its axis.

The form and design of a mandrel. Diagram (1) illustrates the principal features of a mandrel of the plain or solid type. A mandrel is usually made of tool steel, hardened, tempered and ground to size, with a slight taper of .006″ per foot or .0005″ per inch. It is obvious that such a tool would be useless unless the hole into which it must be pressed is reamed accurately to size. The two ends are reduced so that if any burr did develop in use, it would not interfere with the fitting surface of the mandrel. The corners of the mandrel are rounded so that they will not spread when under the pressure of an arbor press or a lead or copper hammer.

The fillet shown is rounded to prevent hardening cracks starting in the corner when hardening. The small end is .0005″ under the nominal size and the large end is easily detected by the fact that the size is stamped on that end. Flats are provided to give a positive drive when the set screw of a lathe dog is pressed against it. The flats can be milled, ground or formed on a shaper and the size stamped on one flat before hardening. The flats can be ground after hardening and the size etched on the large end.

To etch the figures denoting the size of the mandrel. Apply a resist, usually made from quick-drying black asphaltum. When dry, scratch the figures with a scriber and apply one drop of "aqua regia" (hydrochloric and nitric acid). Wash off in water after one minute and clean off the resist with gasoline. This method is suitable for marking any tool of hardened steel.

Mandrel proportions can be obtained by reference to the table of proportions on page 200. Particular attention should be paid to the size and form of the countersink on the ends, as a mandrel must be used over many times and the condition of the work is dependent upon the quality of the centres and the centre holes on which it rotates.

Diagram (2) shows a detail of the centre hole, the 60° countersink is protected by a further countersink, so that if the metal on the end of the mandrel is damaged, the centre hole is untouched and will run true.

The 60° countersink should be large enough to give a sufficient bearing for large work mounted on it and distribute the wear over a greater surface.

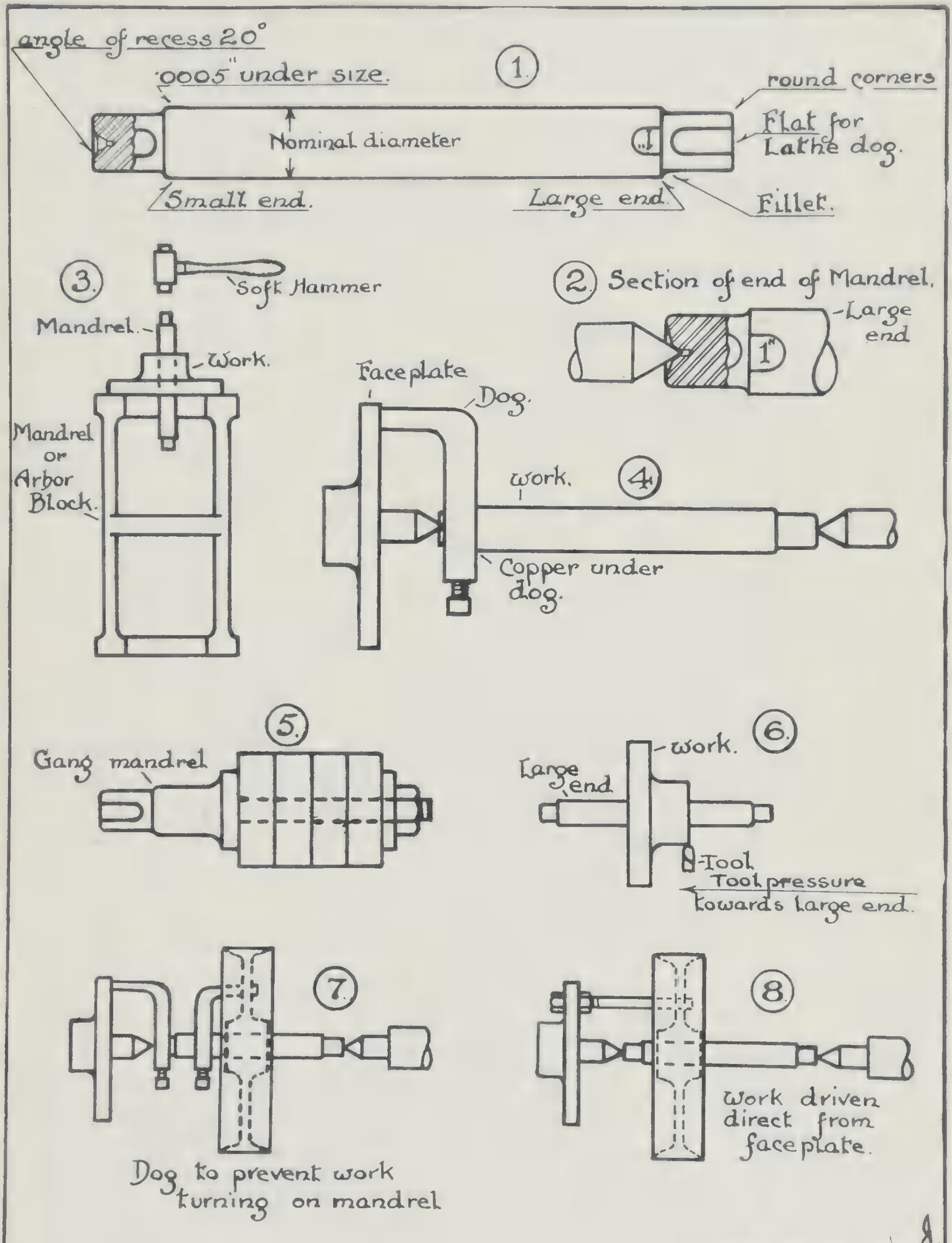
Applying a mandrel to work. Diagram (3). The mandrel is first oiled and the small end placed in the reamed hole, then driven into the work with a lead or copper hammer, while the work is supported evenly by the arbor block.

Turning a mandrel. Diagram (4). The mandrel is turned on centres and made of tool steel which must be hardened and tempered afterwards. It is turned the same diameter throughout, but 1/32″ oversize. This allowance takes care of any warping which may occur in the hardening and gives sufficient allowance for the grinding which follows the heat treatment.

Gang mandrel, Diagram (5), is made the same diameter throughout on the reduced portion and the work is mounted on it and held securely to the mandrel by the pressure of a nut. (Example, gears mounted for cutting on a milling machine).

Direction of cut on work mounted on a mandrel. Diagram (6). Work should be mounted on a mandrel so that the direction of the cut and its corresponding pressure should tend to press the work more tightly on the increasing diameter of the mandrel.

Driving large work which is mounted on a mandrel. When work mounted on a mandrel is driven by the ordinary lathe dog, there is a tendency for the work to slip on the mandrel, because of the difference in the moments—between the mandrel radius and the work radius about which the tool pressure acts. Two methods of offsetting the tendency to slip are shown in diagrams (7) and (8).


angle of recess 20°
·0005" under size.
1.
round corners
Flat for Lathe dog.
Nominal diameter
Small end.
Large end.
Fillet.
3.
Soft Hammer
Mandrel.
Work.
Mandrel or Arbor Block.
2. Section of end of Mandrel.
Large end
1"
Faceplate
Dog.
work.
4.
Copper under dog.
5.
Gang mandrel
6.
work.
Large end
Tool
Tool pressure towards large end.
7.
Dog to prevent work turning on mandrel
8.
Work driven direct from faceplate.

THE MANDREL.

DRILLING AND REAMING IN THE LATHE

The casting should first be snagged on the grinder to remove all irregularities, and it is also advisable to grind at the centre, so that the spotting tool will not have to penetrate the hard skin of the cast iron.

The work should be chucked and tested for true running with a piece of chalk.

Spotting previous to drilling. Diagrams (1), (1A) and (2). If a small drill is used to drill a pilot hole without previously spotting the work, the drill will spring off centre. The spotting tool is rigid and will not spring, so the hole will be started axially true.

A forged tool may be used as shown, or a high speed steel tool bit may be used. The enlarged detail (A) of a spotted hole shows an important feature. The cutting edges of the pilot drill should first strike the sides of the recess so that it will automatically follow the line of least resistance and drill on centre. Combination drills may be used for this work, but they are expensive if broken, and not at all necessary for spotting, as they are specially intended for drilling centre holes for work to be mounted on centres.

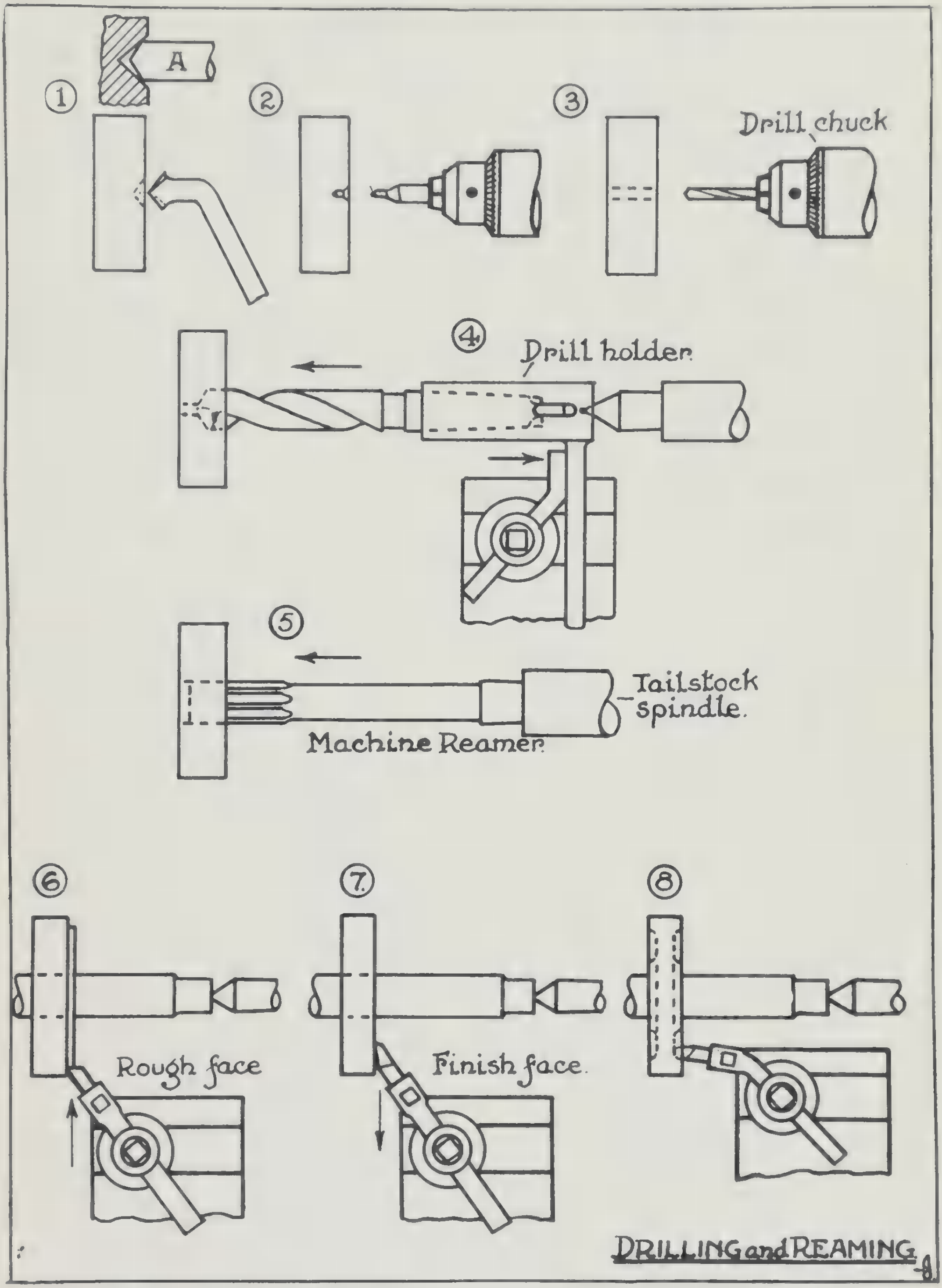
The pilot hole, diagram (3), allows a large drill to cut freely, as the web of a large drill does not cut so freely as the cutting edge of the drill. (See drill grinding, page 25).

Drilling the hole before reaming. Diagram (4). It is essential that the drill be ground perfectly true, otherwise it may drill the hole too large and not leave sufficient metal for reaming.

A drilled hole is never perfectly round and very rarely is it accurate to the nominal size of the drill. For diameters less than 1″, 1/64″ is allowed for reaming. The drill may be held on centres as shown in diagram (4), with the tapered drill shank held positively in a drill holder.

Sometimes drills with taper shanks are fitted to the tailstock spindle, either directly or indirectly, by means of sleeves or sockets. When the drill is held in this manner, only the friction between the taper shank and the tailstock spindle prevents the drill from turning, therefore the method shown in diagram (4) is more satisfactory. If the drill tends to spring at the point, a toolholder in a reversed position may be used, held in the toolpost, in order to push against it and steady it.

Reaming the hole after drilling. Diagram (5). A hand reamer should never be used on a lathe under power. It is better to use a machine reamer fitted with a taper shank to the tailstock spindle, as shown in diagram (5).


A
1
2
3
Drill chuck
4
Drill holder
5
Tailstock spindle.
Machine Reamer
6
7
8
Rough face
Finish face.
DRILLING and REAMING

The work should rotate slowly and should be kept rotating in the same direction until the reamer has been backed out. Finish the hole exactly to size, with a hand reamer if necessary. If care is taken in drilling and reaming, the hole should be perfectly round and accurate to 1/1000″. If the hole is not accurate, it will not fit tight on the mandrel and withstand the cutting pressure of the tool.

Turning work mounted on a mandrel between centres. The mandrel should be so placed on the lathe centres that the tool pressure will tend to force the work more tightly on the mandrel. As far as is possible, the cutting should be in the direction of the headstock of the lathe; therefore the big end of the mandrel should be on the live centre.

Rough facing is shown in diagram (6), with the cutting face of the tool bit at right angles to the hard skin. To finish face, the straight part of the tool bit gives a smooth finish when fed towards the outside of the flanges. (See diagram 7).

Hand tools may be used to scrape the face and fillet to a smooth surface, but must not be used on any face that must be accurate and flat, such as the fitting face of the flange. If gear blanks are machined from cast iron, it is important that the outside diameter be machined very accurately to size, otherwise thick or thin teeth will result, when indexed and cut on the milling machine to the correct depth. Diagram (8) shows the method of turning the fillets in a gear blank.

ECCENTRIC TURNING

This lesson on eccentric turning illustrates the stages of work in the making of an eccentric, similar to the eccentric used in the back gears of a lathe.

Diagrams (1) and (2) illustrate the importance of laying out the work carefully after it has been squared to length. Unless the end lines, which are shown dotted, are in the same plane, the axes from the offset centres will not be in alignment.

To obtain the correct lay out, as shown in diagram (1), the work is mounted in a vee block which rests on a surface plate, as shown in diagram (3). From the centre of the stock, which may be found with a centre square (diagram 6), the scriber of the surface gauge marks the radial lines on the ends and the line joining them. The offset is now measured, as shown in the end view in diagram (4).

Mark the centre on each end with a centre punch and drill one centre hole on the axis of the stock at each end. Mount on centres and turn to diameter as shown in diagram (5). Check the diameters of the work at each end carefully. If not correct, the lathe centres are out of alignment, and this must be corrected before the finished diameter of work is reached.

To check offset. Diagram (7). Mount the work again on centres, but with the centres lightly bearing against the second offset centre punch recesses. The graduations on the cross feed screw collar are used to measure the offset; and paper is used to measure the pressure between the tool point and the work.

Diagram (B), number 9, shows the tool bit in contact with the work when the throw is towards the tool bit. Turn the cross feed handle to the right and use paper to measure the pressure in this position. Now swing the work until the throw is away from the tool bit, as shown in diagram (A) number 9. Now move the tool in by turning the cross feed screw until the tool contacts with the paper between it and the work. (B)

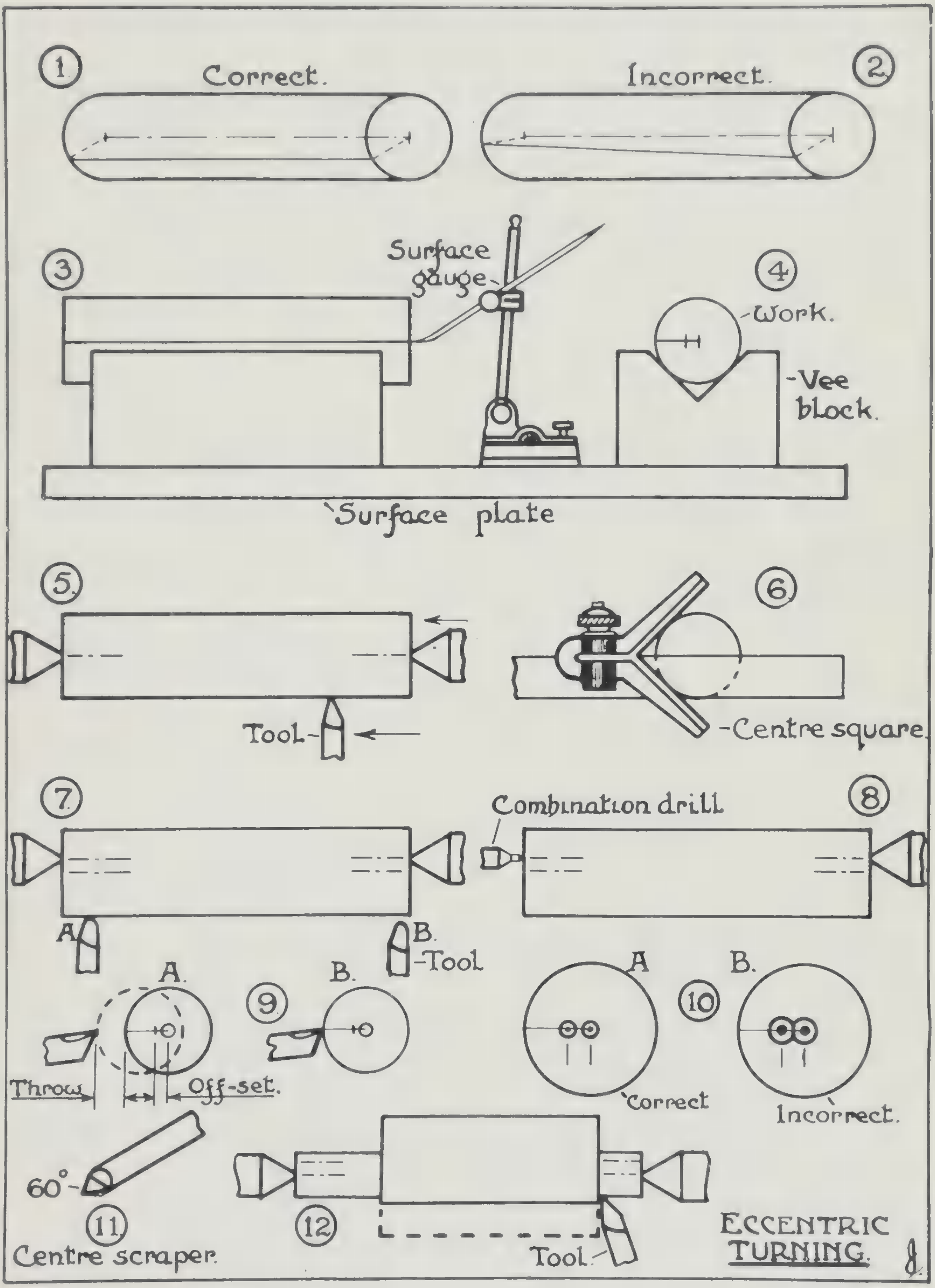
Note the number of divisions on the cross feed screw collar, and this will be the measurement of the throw. Make this check at both ends of the work. Then if correct, centre the work with a combination drill by hand, as shown in diagram (8).

Drilling the centre holes. Diagram (10). The centre hole countersinks must be made as shown at A, diagram (10)—and not as shown at B—otherwise the centre holes will wear towards each other and reduce the offset. If the offset is not correct after drilling, the second

centre hole, a scraper, as shown in diagram (11), may be used to correct the offset. This must be followed by countersinking to make the countersink again round.

Turning the ends on the offset centre. Diagram (12). Many difficulties may be encountered here unless the operator appreciates all the fine points of tool grinding, tool setting, and the reactions of the work to the angles of the tool bit.

First, the irregular cut causes an intermittent load on the centres, the tool bit and the tool holder. The strain on the centres can be kept to a minimum by presenting the cutting face of the tool bit in a position almost at right angles to the axis of the work, but slightly undercutting, so that the action of the cross feed screw can square the face. To obtain a smooth surface on the periphery of the work, round the point of the tool slightly and do not have too much contact with the work, otherwise the tool will tend to dig in. (Read Tool Grinding and the Mechanics of Lathe Tools, pages 123, 179).


1
Correct.
Incorrect.
2
3
Surface gauge
4
Work.
Vee block.
Surface plate
5
6
Tool
Centre square
7
Combination drill
8
A
B.
Tool
A.
B.
9
10
A
B.
Throw
Off-set.
Correct
Incorrect.
60°
11
Centre scraper
12
Tool
ECCENTRIC TURNING.

LATHE GEARING

A simple form of lathe gearing is illustrated here to show the connection between the lathe spindle and the lead screw.

The reverse gears. The rotation of the lead screw of a lathe is prevented by disconnecting the gear drive through the reverse gears. The remaining gears in the train remain set up for use when required. Disconnecting the gears when not in use is a saving in wear of the gear teeth and reduces the noise made by the rotation of the gears. On some lathes when the lead screw performs a double function, that of a lead screw and feed rod, there is also a danger that the screw will grip any loose clothing because of the sharp edges where the spline cuts through the threads.

The reverse lever is usually pivoted on the stud and carries the reverse twin gears, which are always in mesh with each other.

Diagram (1) shows both twin gears in the drive between the spindle gear and the stud gear. This makes 4 gears, an even number; consequently the stud gear rotates in the opposite direction to the spindle gear.

Diagram (2) shows only one twin gear in the drive. This gives an odd number with the same direction of rotation between spindle and stud gear.

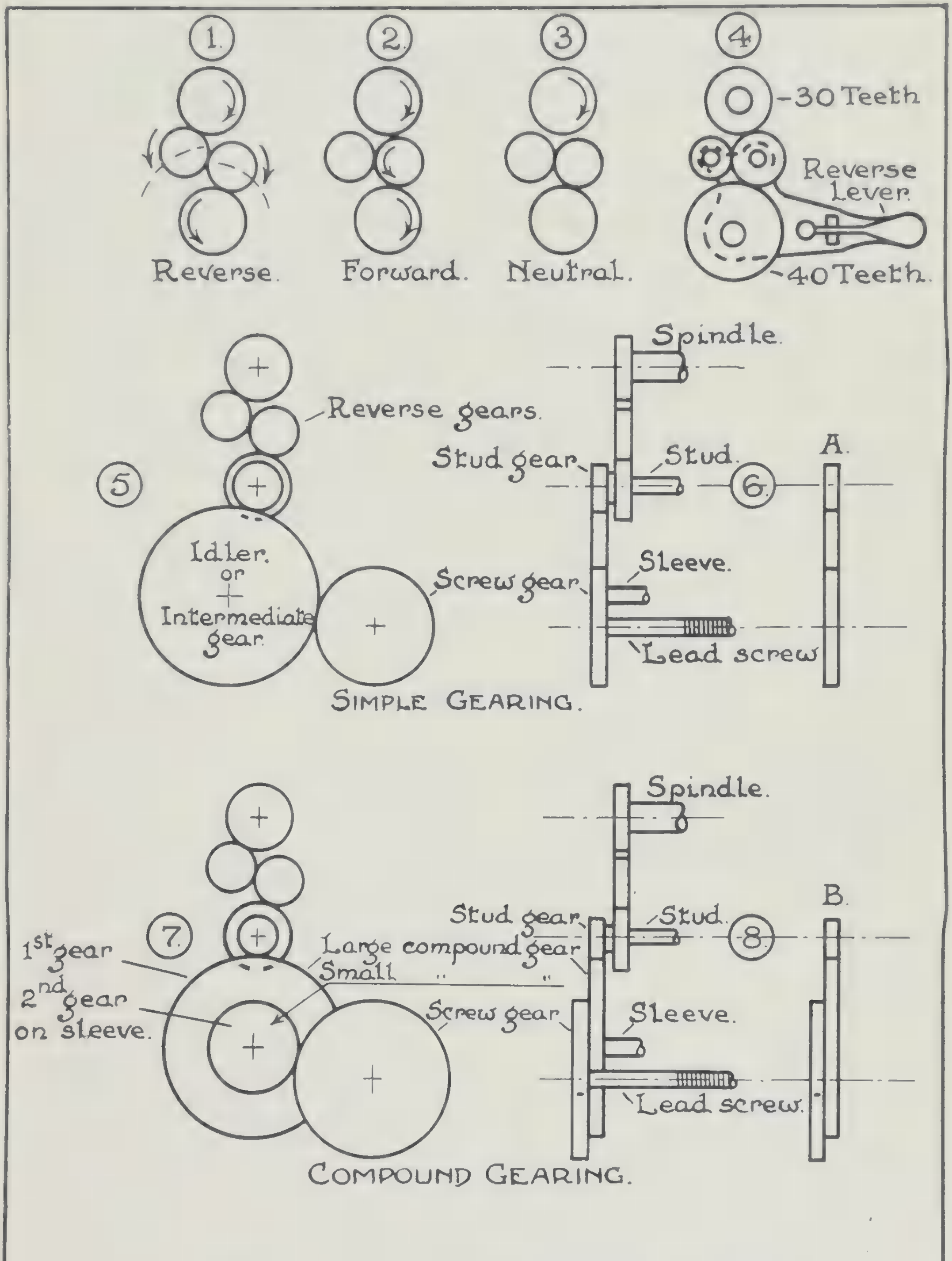
Diagram (3) shows the neutral position between spindle and stud gear, with no rotation of the stud gear.

Ratio of spindle and stud gear. If the spindle and stud gear have the same number of teeth, they will rotate equally. This is important in working out the gear ratios between spindle and lead screw, as it is only necessary to figure the gear ratio from the stud to the lead screw if the stud and spindle ratio is 1 to 1.

Diagram (4) shows an arrangement sometimes seen on a lathe where the spindle gear and the stud gear have a different number of teeth. This, of course, varies the ratio and rate of rotation of the gears. In the diagram shown, if the spindle makes one revolution, the stud gear makes $\frac{3}{4}$ of a revolution.

The rate of rotation of the stud is inversely proportional to the number of teeth, that is, while the spindle gear makes 4 revolutions the stud gear makes only 3. This must be taken into account when figuring the ratio between spindle and lead screw.

Simple gearing. Diagrams (5) and (6) show two views of a lathe geared up with a simple train of gears. The stud and spindle ratio is



LATHE GEARING.

1 to 1 so that the effective ratio gears in the train are shown at A, driving direct, the ratio being governed by the first and last gear, as the intermediate gear functions only as an idler. An idler gear connects other gears and only varies the direction of rotation. Usually on most lathes, if the spindle and lead screw ratio is less than 1 to 5, a compound train is used.

Compound gearing, (diagrams (7) and (8),) is used when the centre distance between the stud and lead screw prevents a greater ratio than 1 to 5. The pinion placed on the stud drives the first gear on the sleeve. The second gear on the sleeve rotates equally with the first gear on the sleeve and connects to the gear on the lead screw. All these gears affect the ratio between the stud and lead screw and consequently between the spindle and lead screw.

Diagram B shows the difference between a compound and a simple train, as at A above, there being two distinct lines of drive in a compound train. Idlers could be used to connect gears in a compound train, but none are shown here.

It is important to note that diagrams A and B show the removable or changing gears, and diagrams (1) to (4) the fixed gears.

CUTTING A U.S.S. THREAD

To cut a smooth and accurate thread in a lathe, depends upon the operator's ability to recognize the various influences which have an important action on the tool when cutting. A beginner does not usually pay enough attention to the small things, and, in consequence, does not produce nice work.

The various forces acting on the tool bit are of vital importance and cause a peculiar action on the cut, due to, (1) the manner in which the tool is ground, (2) the direction of the cut, (3) the position and type of the toolholder, (4) the fit of the slides of the machine and the use of proper cutting compound and suitable cutting speed.

Diagram (1) shows that when the tool cuts on both sides of the thread, there are reactionary forces acting at right angles to the cutting faces, which have a tendency to pinch the tool bit and draw it into the work. This is a common source of many difficulties and if other errors in tool setting allow, it will cause disruption of the metal being cut and produce a rough surface on the thread.

Side clearance of the tool. Diagram (2) shows the front elevation of a tool bit in the V groove of the thread. The tool shown has too much side clearance and will tend to dig into the surface of the thread, producing a rough cut.

Diagram (4) shows a tool with correct side clearance, sufficient to cut without the heel of the tool rubbing the work. The cutting edge of the tool has ample support and plenty of metal for dissipating the heat caused by cutting; at the same time, it will not dig into the surface and will produce a smooth cut. The side clearance should be sufficient to clear the metal at the root of the thread, because the root has a greater inclination than the top of the thread.

Front clearance. Diagram (3) has an absurd amount of front clearance, even if incorrectly set above centre, as shown, the tool has a weak cutting point and would tend to wear quickly and dig into the thread. Diagram (5) shows the correct amount of front clearance from 10 to 15 degrees. This is a very important item in tool grinding for threading, and, if followed, will produce a smooth thread.

The height of the tool bit. Diagram (5) shows the correct height for a tool bit when cutting a thread. Because of the gripping action between the tool bit and the work, the tool should be set on centre so that if the toolholder springs from any cause when cutting, it will be pushed down by the chip pressure, away from the work. If the tool were set

above centre, it would be pressed down and into the work, producing a heavy cut, which would probably spoil the thread.

Feeding the tool into the work with the cross feed screw. Diagrams (6) and (7). If this method of cutting is adopted, the point of the tool which is the weakest part of it, receives the full force of the cutting action all the time. In addition to this, two chips converge on each other from both sides of the thread and the resistance they offer to each other, in coming away from the cutting faces, gives extra and unnecessary burden on the tool which can be felt in the cutting action of the machine.

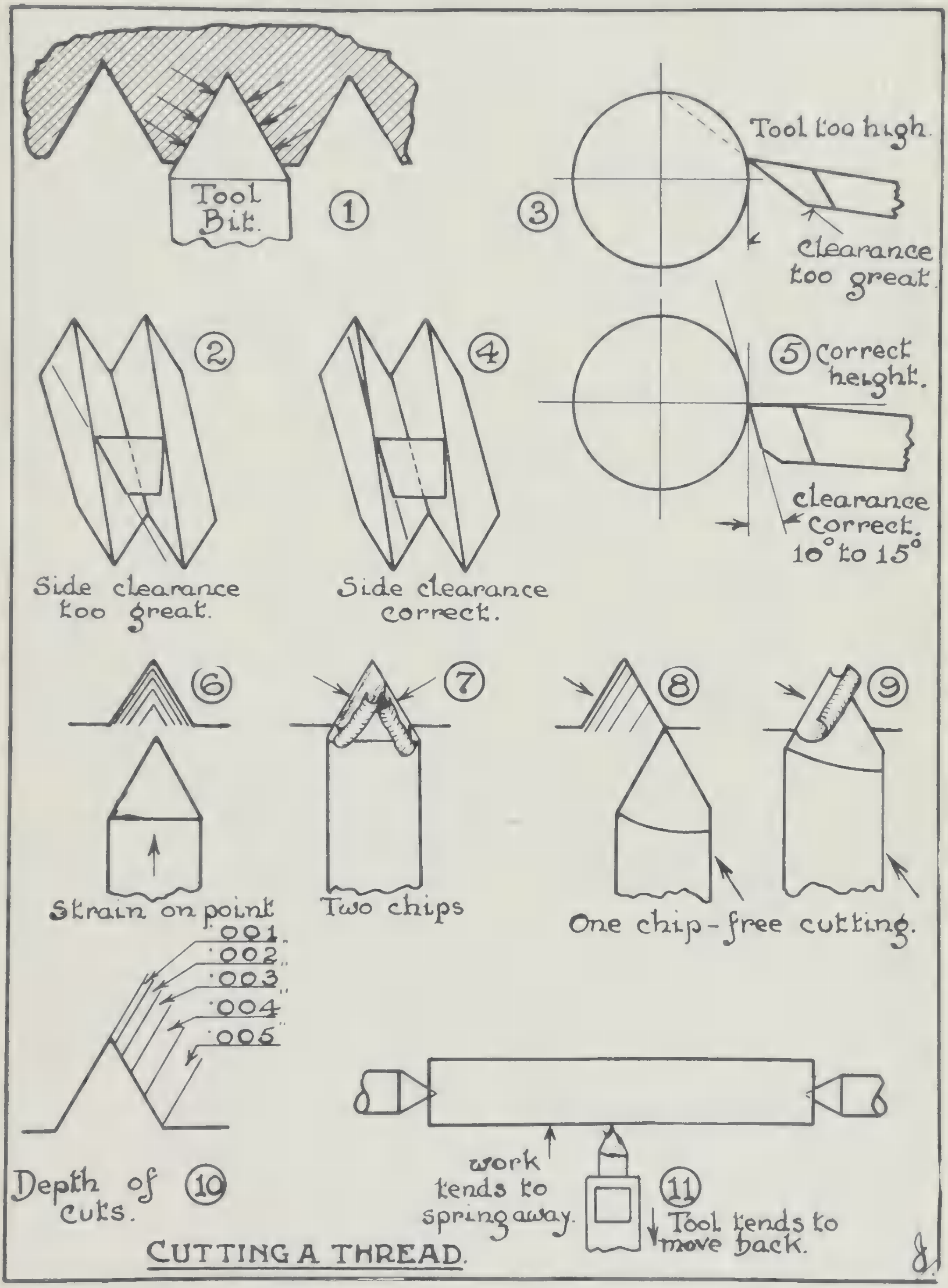
Feeding the tool into the work with the compound slide rest set at 60 degrees. Diagrams (8) and (9). By this method, all the bad features shown in diagrams (6) and (7) have been eliminated. The strain of cutting is distributed evenly over one cutting edge. The single chip produced has ample freedom in coming away from the cut, and the binding action of the V on the tool is reduced to a minimum. If the carriage is held back by slight pressure on the hand wheel, the right side of the thread can be smoothed on the last few cuts.

Depth of cut when threading. Diagram (10). As the cut increases in width, due to the increased depth in the V, the depth of cut should decrease as shown. At intervals, a free cut should be made to take care of any spring in the machine. The space between the lines represents the amount of the depth of cuts indicated, and the lines represent the time when free cuts should be taken.

Spring of the work and machine. Diagram (11). The work resists penetration of the tool and tends to spring away from it. On the other hand, due to the chip pressure, the tool is pushed away from the work if any slackness in the fitting of the machine allows it. This is the reason why free cuts should be taken to produce a smooth surface on the thread, particularly when the tool is well down in the V near the finish of the thread.

Use of cutting compound. The proper selection of a suitable cutting compound for different metals is necessary if a nice thread is required. This point can easily be proved by cutting one thread with a proper compound and one **without.**

These points are intended for beginners in thread-cutting, and if slow cutting speeds are used at first, a first-class job will result.


Tool Bit.
1
3
Tool too high.
Clearance too great.
2
4
5 Correct height.
Clearance correct. 10° to 15°
Side clearance too great.
Side clearance correct.
6
7
8
9
Strain on point
Two chips
One chip-free cutting.
.001"
.002"
.003"
.004"
.005"
Depth of cuts.
10
work tends to spring away.
11
Tool tends to move back.
CUTTING A THREAD.

SETTING THE THREAD TOOL FOR SCREWCUTTING

If proper fitting threads are to be obtained, the operator must exercise care in the grinding of the tool bit, the selection of the toolholder, the position of the toolholder in relation to the axis of the work, and the amount it overhangs from the toolpost.

Grinding the tool bit. First, the type of thread must be known. If it is U.S.S. thread, the profile of the tool will be a 60° angle. It is not sufficient, however, just to grind this angle on the tool bit, but one must know how the toolholder will be placed when the tool bit is correctly placed with regard to the axis of the work. As far as possible, the toolholder works best when it is slightly beyond the right angle position with the work axis, that is, if undue pressure is exerted on it, it would not spring or turn into the work to increase the depth of the thread.

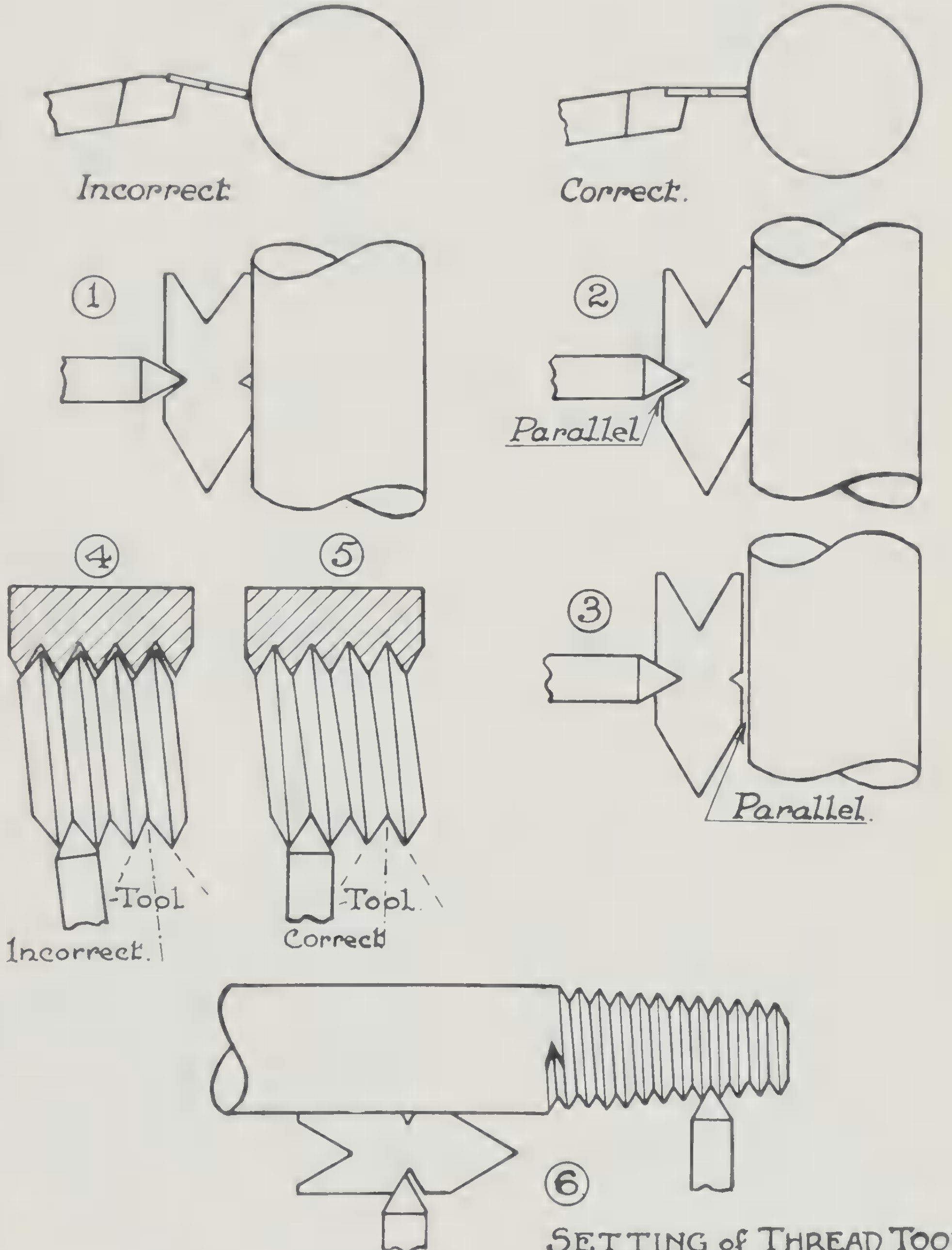
The amount of flat on the end of the tool for any given thread should be tested with a screw pitch gauge of the desired pitch. (See page 59).

Right or left hand tools. If the thread being cut is right handed, the side clearance of the tool bit will be chiefly on the left side of the tool, looking towards the work. If the thread being cut is left handed, the side clearance will be on the right side. The top face of the tool, when in position in the toolholder, is generally horizontal and exactly on centre. If the compound slide is used to increase the depth of cut, the tool is sometimes given a side rake. The right hand tool in this case would have a side rake from the left or cutting side of the tool downwards, and the left hand tool would have a side rake from the right hand cutting side downwards. The tool, before using, should be rubbed with a fine abrasive stone, as a thread can only be as smooth as the edge of the tool that cuts it.

Setting the tool to the work with a thread gauge. Diagram (1) shows the plan and elevation of a tool being set with a centre or thread gauge with an angle of 60°. These diagrams show the thread gauge inclined from the horizontal in an incorrect position, so that the angle is more than 60°. It is necessary, therefore, when using a thread gauge to set it correctly, as shown in diagrams (2), in a horizontal position.

One method of checking the tool position is shown in diagram (2). Here the gauge is held tight against the work and the parallel space between it and the tool bit is checked.

Diagram (3) shows another method. Here the gauge is held tight against the tool bit and the parallel space against the work is checked.



SETTING of THREAD TOOLS.

The effect of incorrect setting of tool with the axis. Diagram (4) shows a tool which has cut a thread while incorrectly set, that is, the line that bisects the profile angle of the tool (60°) is not at right angles to the axis; consequently, when fitted into a correctly cut thread as shown, the threads touch and wear at the points only, and will not fit together at all, unless they are a very poor, slack fit, as shown. This condition is very bad. A tool correctly set and a thread correctly cut are shown in diagram (5), giving a good fit and maximum wearing surface and strength.

Setting tool for cutting threads on tapered work. The thread tool should be set square with the axis of the work, as shown in diagram (6). The most satisfactory way to cut such a thread is to use a taper attachment, so that the work runs on the centres in alignment, and the tool follows the taper as set on the taper attachment. If an attachment is not available, the work may be set over from the tailstock to give the correct taper, but the tool setting is still the same as shown in diagram (6). If work is held in the chuck, the tool must be backed out slowly by hand to give the correct taper and form of thread; but this is not very satisfactory, if particular work is desired.

THREADING TOOLS

Thread cutting is in reality a forming operation, the form of the tool depending upon the type of thread being cut. The formed shape is cut like a helix around the cylinder, and as is well known with ordinary forming, there is a tendency to produce chattered work. Because of the pinching action of the sides of a thread on a threading tool, and because of the tendency of the tool to dig in, many tool holder manufacturers have designed special threading toolholders for ordinary screw-cutting in a lathe.

For production work, special threadcutting die heads have been designed with chasers such as those produced by "Landis" and "Geometric" and are excellent tools, but on account of their cost, are only used for large quantities of work.

Spring threading tool holder ("Agrippa" patented), as shown in Diagram (1). This toolholder is designed on the "goose neck" principle. The metal is so weakened at the point (A) that any overload at the cutting edge will cause the tool to spring about the pivot point (A). It is obvious that the cutting edge of such a tool should be set on centre with the work. If it were set above centre, it would produce worse results than a solid toolholder, because the tool bit would readily spring down into the work. If it is set carefully on centre, the tool bit will spring away from the work and produce a smooth thread. The cam at the point (B) may be adjusted to take out all the freedom for springing for rough cuts or to allow freedom in any required amount for light finishing cuts.

Chaser type of toolholder. ("Pratt and Whitney" patented). Diagram (2). The chief advantage of this toolholder is that the operator has a tool ready ground for use. Several chasers are provided for different numbers of threads. When the toolholder has been held rigidly in the tool post in approximately the correct position for height, a very fine adjustment for height can be obtained by turning the screw at (A), after unlocking the chaser clamp at (B). When grinding the tool to sharpen, it is only necessary to grind the top face, and lines are provided on the chaser to guide the operator and retain the correct rake angle. The front clearance angle is established by grooving the chaser into the toolholder.

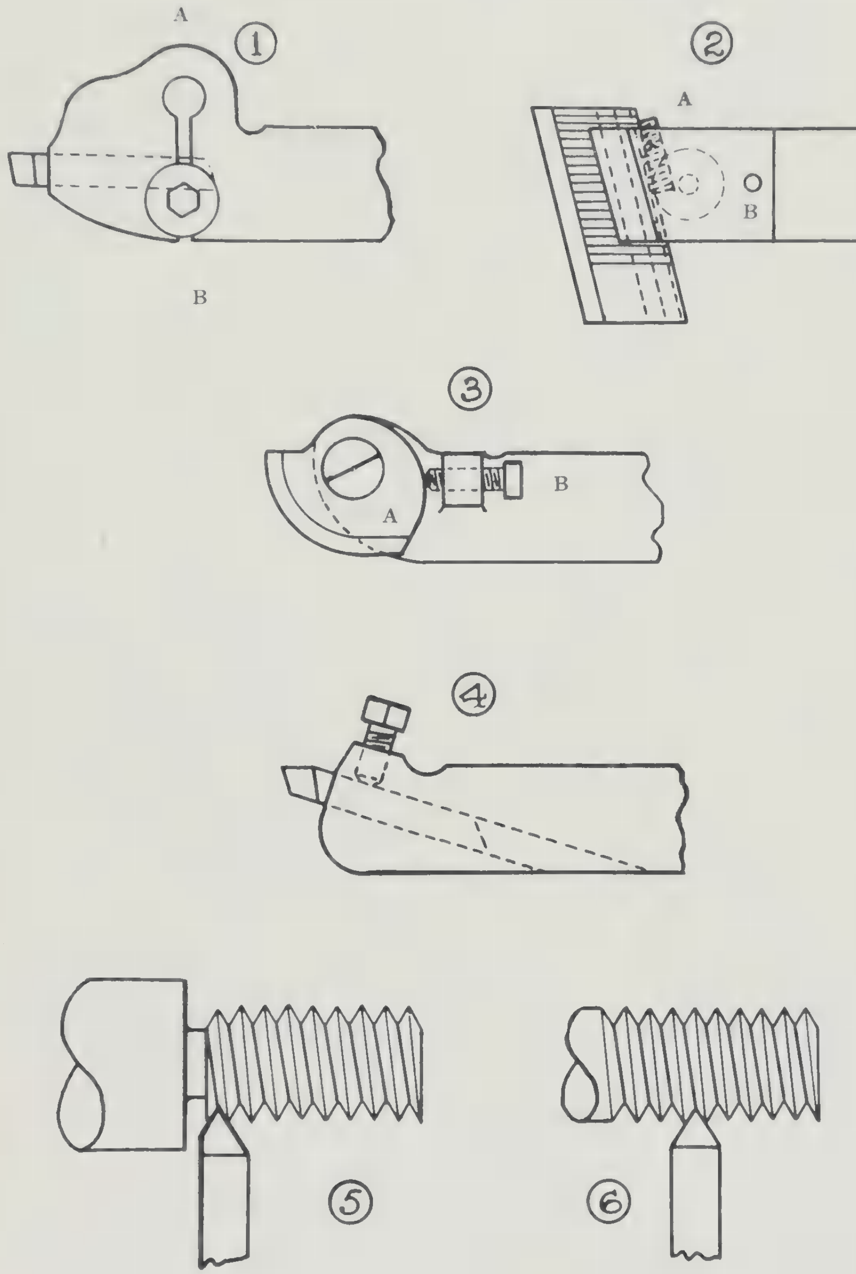
Pivot formed threading tool holder. ("Armstrong" patented). Diagram (3). This tool, like the one illustrated in diagram (2), has a formed tool bit for different number of threads. To adjust the cutting edge, the screw (A) is loosened and the formed tool turned around by

the set screw (B). This adjustment may be used to change the height of the tool bit or slightly vary the rake angle. To sharpen the tool, only the top face is ground.

Plain threading toolholder. Diagram (4). This is the common toolholder that is used for most operations of lathe turning. When used for screw cutting, the tool bit must be ground by the operator to the correct cutting angle, and profile and the top will have to be ground to offset the inclination provided by the toolholder, and to give it the correct rake for screw cutting, which is usually horizontal. The tool can be adjusted for exact height by moving the toolholder or by loosening the set screw that holds the tool bit, and then moving the tool bit up or down the inclined slot as desired. This latter method is better because it does not change the position of the tool at right angles to the axis of the work.

Threading to shoulder. Diagram (5). Most special tool holders for threading are made of the offset type, so that one can thread close to the shoulders or thread work held in the chuck without any difficulty. The tool bit should be ground offset as shown in diagram (5), so that the form of the thread can be completely cut before the side of the tool bit strikes the shoulder of the work. A groove should always be provided as shown, cut to a depth equal to the pitch of the thread so that the point of the tool bit will have clearance.

Threading work without shoulders. Diagram (6). It is much easier to thread such work if a clearance is provided for the tool to run in at the completion of the thread cutting, but if the thread must stop, as shown, the work should be turned by hand for the last revolution, and the tool backed out carefully; otherwise the tool bit will be broken by striking an irregular amount of metal. For this class of work, the point of the tool is best ground in the centre.



THREADING TOOLS.

MEASURING THREADS

It is very important that threads be measured accurately, because a few thousandths of an inch will make a great difference in the fitting of the threads. The shape of the tool bit that cuts the threads, the amount of flat on the end of the tool and the diameter of the work influence largely the final fit.

U.S.S. thread tool gauge. Diagram (1). This tool is a template for testing the correct angle and amount of flat of the tool bit being ground preparatory to thread cutting. The V at A tests the 60° angle and each of the other V grooves is marked for the number of threads per inch that the tool being ground is intended to cut.

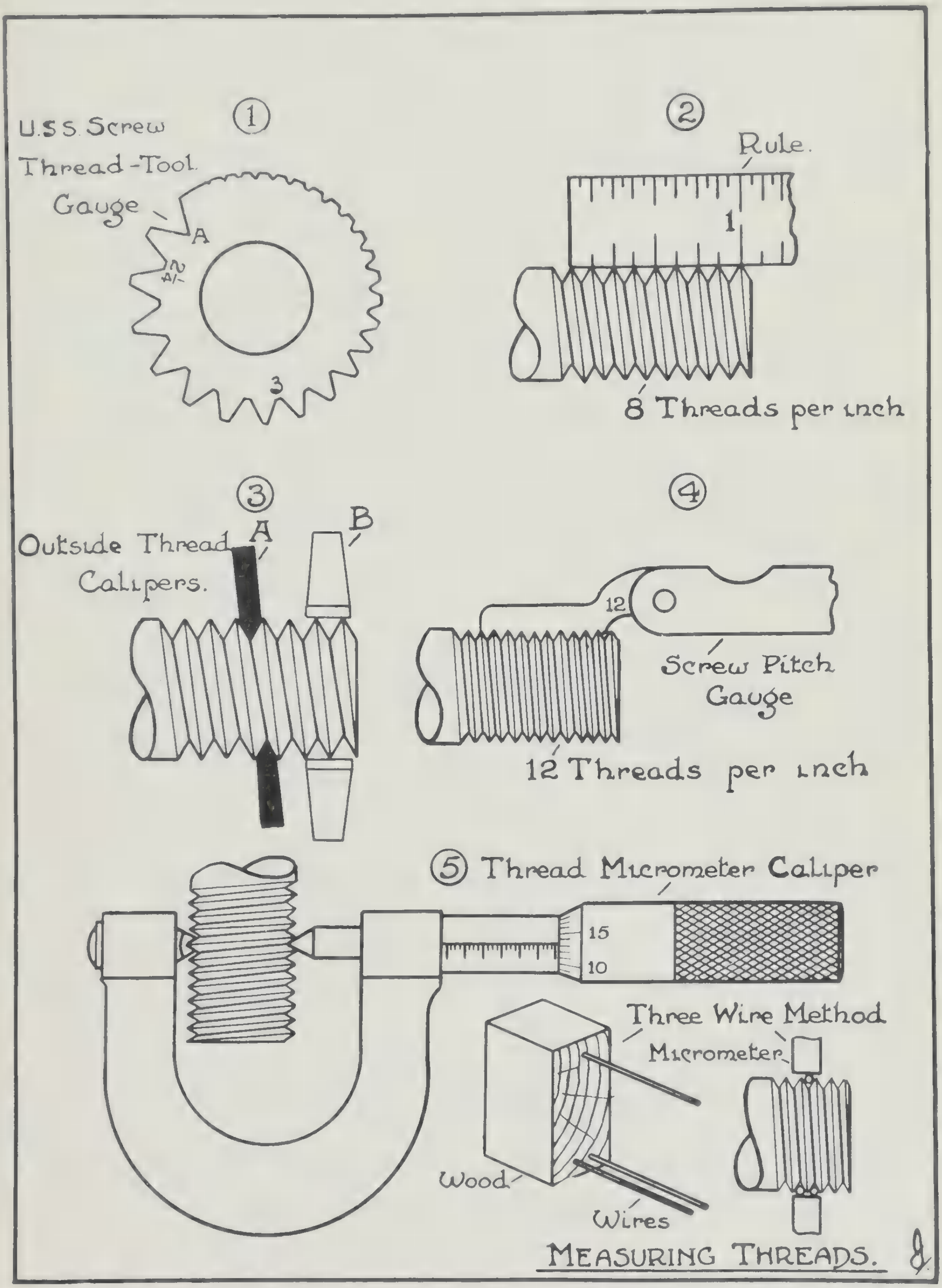
Testing number of threads per inch on work. Diagram (2). When testing a piece of work that is threaded, place a rule on edge, as shown, with one inch mark corresponding to the top of one of the threads. Count the number of V grooves or spaces that are included in one inch, and that will be the number of threads per inch. Metric threads can be measured so many per centimeter, although the base of this system is one millimeter and a metric screw pitch gauge should be used. Sometimes it is advisable to check the number to a 2″ space. Example, 4½ threads per inch would be exactly 9 for a 2″ length.

Measuring threads with calipers. Diagram (3). The outside thread calipers shown at (A) are tapered at an angle so that they will measure only the diameter of the work at the root or bottom of the thread. The outside thread calipers shown at (B) are made wide enough so that they will bridge across the tops of the threads to measure the outside diameter. Most beginners make the outside diameter of threaded work larger than the nominal size. It should be obvious that if a plain plug cannot enter a plain hole if the plug is oversize, an oversize threaded piece cannot enter a threaded hole. Measure the work carefully as a few thousandths of an inch make a great difference between a proper fit and a poor fit.

Use of the screw pitch gauge. Diagram (4). This tool may be used to test work to find how many threads per inch. This is done by trial, estimate the number of threads approximately, then try the leaves of the gauge near that number until one is found to fit perfectly.

The most important use of this tool is to test the threads of work being cut in the lathe. If the outside diameter of the work is correct and the screw pitch gauge fits the work perfectly, the thread is finished and should fit the corresponding nut.

The leaves are made narrow in width to allow them to fit inside small threaded holes to test the correct cutting of the thread. Some screw


1
U.S.S. Screw
Thread-Tool
Gauge
A
3
2
Rule.
1
8 Threads per inch
3
Outside Thread
Calipers.
A
B
4
12
Screw Pitch
Gauge
12 Threads per inch
5 Thread Micrometer Caliper
15
10
Three Wire Method
Micrometer
Wood
Wires
MEASURING THREADS.

pitch gauges have stamped on them the double depth of the thread corresponding to the pitch of the thread marked on the leaf.

Screw thread micrometer calipers. Diagram (5). In this micrometer, the anvil is hollowed to fit the thread and the spindle is pointed to fit the corresponding V, so that when they fit together, O is registered on the thimble corresponding to the line. When fitted to test a screw the diameter measured is the pitch diameter or the outside diameter, less the depth of one thread. Example. Caliper reading or pitch diameter for U.S.S. threads $D - \frac{.6495}{N}$ Look up table, page 201.

Example. For 1″ diameter, 8 threads per inch, pitch diameter is given .9188″. To test a 1″—8 U.S.S. spindle, the thread micrometer would measure .9188″ for correct pitch diameter.

To test diameter of threaded work with ordinary micrometer by the *"Three wire method"*. Three wires are used as shown, held in a wood block. They are of equal diameter, but any diameter smaller than the pitch of the thread will do, providing they project beyond the outside diameter of the work. Place the wires in the grooves as shown, and use the micrometer against them to get the reading for correct depth of cut.

To test U.S.S. form of thread. To the diameter of the screw, add three times the diameter of the wire and from the sum subtract the quotient obtained by dividing 1.5155 by the number of threads per inch. Micrometer measurement across work and wire M, diameter of wire W, diameter of work D, number of threads N. Formula :—

$$M = D + 3W - \frac{1.5155}{N}$$

Example. To measure 1″—8 U.S.S. thread, wire selected .072″ diameter.

$$3W = 3 \times .072'' = .216''$$

$$\frac{1.5155}{N} = \frac{1.5155}{8} = .189$$

$$\text{Measurement} = 1.000'' + .216'' - .189 = 1.027''.$$

To measure a V thread, the constant $\frac{1.732}{N}$ is used.

To measure a Whitworth thread, the constant $\frac{1.6008}{N}$ is used, and 3.1657 times the diameter of the wire as in the above rule.

THE USE OF A THREAD DIAL

A thread dial is used on a lathe to save time in screw cutting and to guide the operator in engaging the half-nuts at the proper place with the lead screw, in order that the thread tool may engage in the previous grooves cut in the work.

Even geared lathe. A lathe is even geared when the number of teeth on the spindle gear equals the number of teeth on the reverse spindle stud. These gears are fixed on the machine and are connected by the reverse gears.

Threads cut without thread dial. If the lead screw of a lathe is 8 pitch, the half-nuts may be engaged at any time to cut threads on the work the numbers of which are multiples of the number on the lead screw, such as 8, 16, 32, 40 threads, etc.

Note. This is only true when the lathe is "even geared". If 4 threads per inch on the work were being cut and the half-nuts were engaged at any time, the tool would cut on the thread correctly or split the thread exactly in two and spoil the work. This introduces the advantages of using a thread dial.

The thread dial may either be permanently fixed on the carriage of the lathe or may be engaged when required, to prevent undue wear on the worm gear. The lead screw of the lathe engages with the worm gear of the thread dial and is connected by a spindle to the dial face which rotates past the stationary zero mark.

Diagram (7) shows a thread dial, the worm gear of which has 32 teeth and the lead screw has 4 threads per inch and is single-threaded. Consequently, every time the lead screw revolves once, the worm gear and dial revolve 1/32 of a revolution. For every revolution of the worm gear and dial, and the lead screw revolves 32 times.

Diagram (8) shows an enlarged view of the dial in plan. It has 8 main divisions and 16 sub-divisions. If the half-nuts were engaged and the dial made 1 complete revolution, the lead screw would have made 32 revolutions; and because the lead screw is single-threaded and $\frac{1}{4}''$ pitch the carriage would have travelled 32 x $\frac{1}{4}''$ which equals $8''$. If, therefore, there are 8 main divisions on the dial, for each division the carriage would travel $\frac{1}{8}$ of $8''$, that is $1''$.

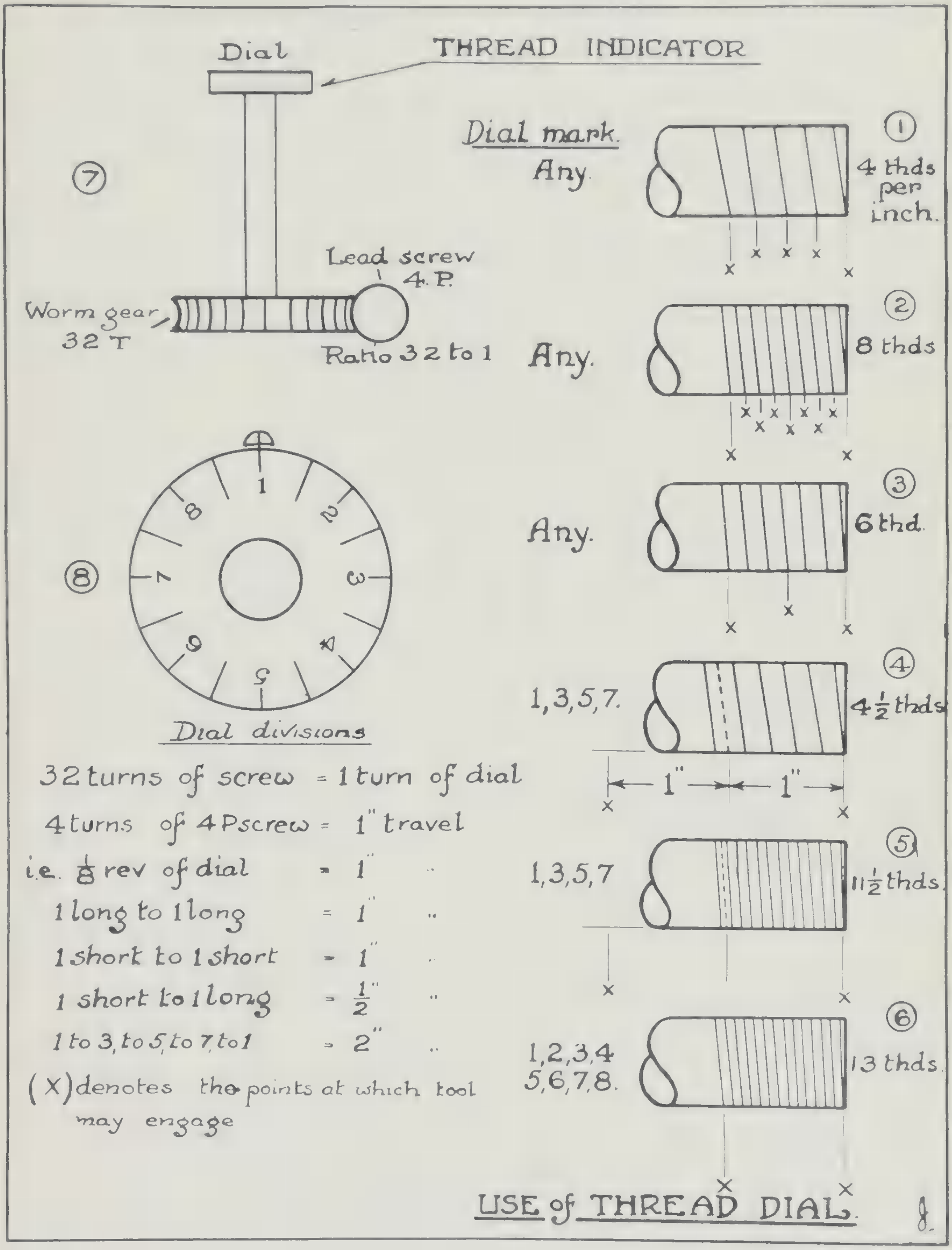
Note. It is well for a lathe operator to understand what the divisions of a dial represent by applying the above method, because of the varying types of thread dials used.

Graphical examples for engaging threads are shown in Diagrams (1) to (6). Diagram (2) shows that the half-nuts may engage at any place, because there are 8 threads per inch on the work, the same as the lead screw. Diagrams (1) and (3) show the thread dial must be used by engaging the half-nuts when any of the 16 divisions pass the zero mark, representing ½″ travel of the carriage.

Diagram (6) shows that only the numbered divisions on the dial may be used, representing 1″ travel of the carriage.

Diagrams (4) and (5) show that every alternate division must be used representing 2″ travel, that is numbers 1, 3, 5 and 7, or 2, 4, 6 and 8. When engaging the half-nuts, watch the dial and lift the half-nut lever easily when the desired marks coincide, to prevent undue wear on the half-nuts. A lathe fitted with a reverse or backing belt may be used if there is no thread dial provided with the machine. In this case the half-nuts are engaged all the time until the job is finished, but the thread tool must be backed out of the thread being cut, to prevent spoiling the thread on the return stroke.

This method is very slow but suitable for short threaded work and is not nearly so effective on long work as using a thread dial.


THREAD INDICATOR
Dial
⑦
Lead screw 4. P.
Worm gear 32 T
Ratio 32 to 1
⑧
Dial divisions
32 turns of screw = 1 turn of dial
4 turns of 4 P screw = 1" travel
i.e. 1/8 rev of dial = 1" ..
1 long to 1 long = 1" ..
1 short to 1 short = 1" ..
1 short to 1 long = 1/2" ..
1 to 3, to 5, to 7, to 1 = 2" ..
(X) denotes the points at which tool may engage
Dial mark.
Any. ① 4 thds per inch.
Any. ② 8 thds
Any. ③ 6 thd.
1,3,5,7. ④ 4 1/2 thds
1" 1"
1,3,5,7 ⑤ 11 1/2 thds.
1,2,3,4 5,6,7,8. ⑥ 13 thds.
USE of THREAD DIAL.

CUTTING A SQUARE THREAD

A square thread is used to transmit motion in a direction parallel to its axis, such as can be found in the function of a screw-jack.

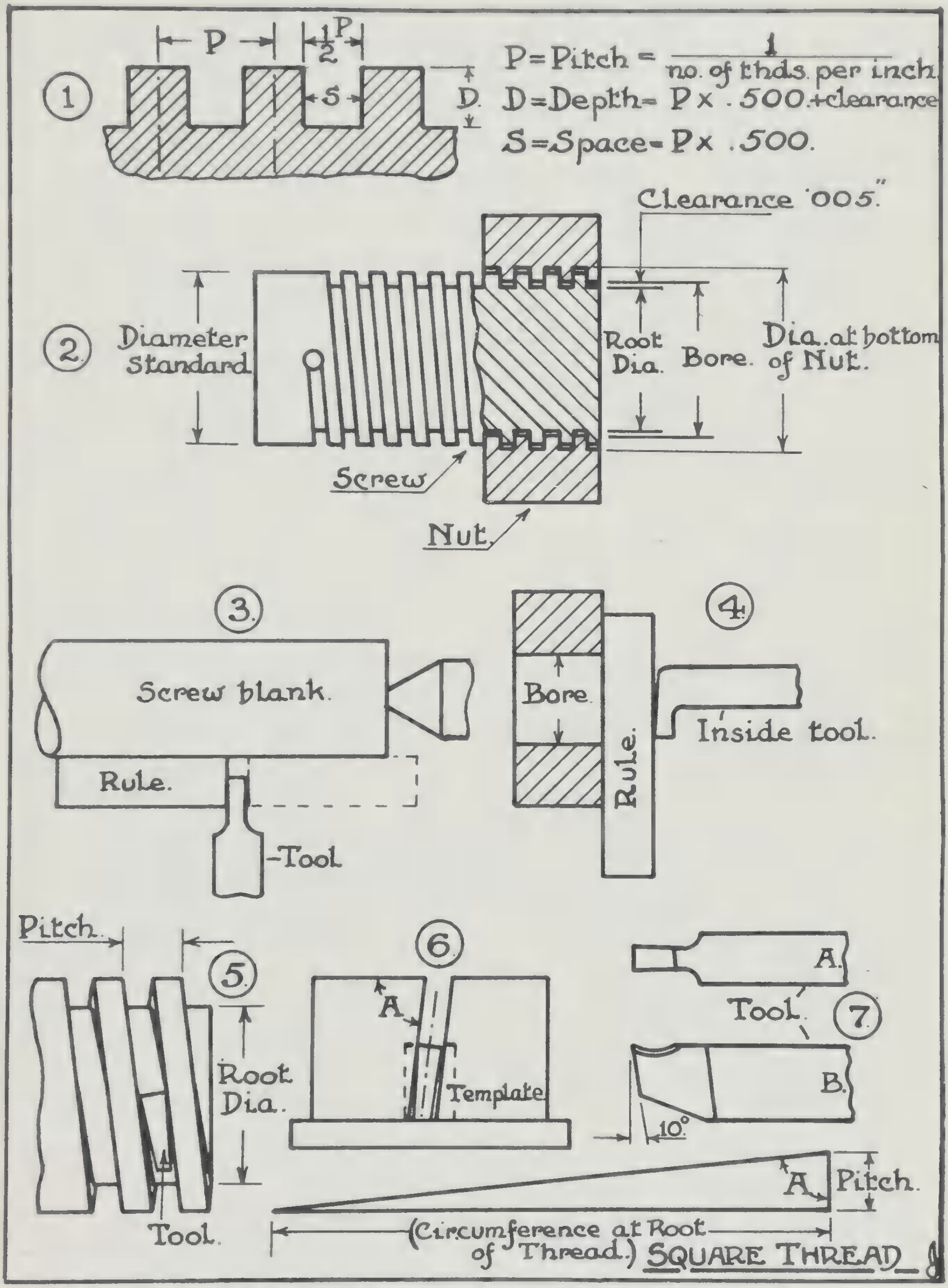
Diagram (1) shows a section of a square thread. The space is nominally one-half the pitch, or equal in width to the thread itself, but as any lathe operator knows, if this were exactly so in the screw and nut, they could not fit together. In practice, therefore, one must be slightly smaller than the other to allow for a running fit. Usually there is sufficient "back lash" between the half-nuts and lead screw to make the space slightly larger than the thread. It is very important that proper clearance be made at the bottom of the thread. Although .005″ is sufficient, beginners are advised to make this .01″, as it is common to find interference between nut and screw through not providing proper clearance.

Diagram (2) shows a section of a square thread of a screw and nut in mesh. The drawing shows the hole that was drilled in the screw before starting to cut the thread, to allow clearance for the tool on the termination of the cut. The bore of the nut before threading should be equal to the diameter of the screw minus the pitch of the thread.

Setting the tool to cut a square thread on a screw. Diagram (3). This illustrates the plan or profile of the tool being set in position at right angles to the axis of the work, preparatory to starting the cut. The profile of the tool shows clearance back from the cutting edge. When the rule is placed against both sides of the tool, as shown, the clearance angle can be compared.

Cutting an internal square thread. Diagram (4). This illustrates the plan of the boring or inside threading tool being set in position. The rule placed against the face of the work is used to check the position of the tool at right angles to the axis of work. It is important to obtain a correct thread which will fit the faces of the threads of the screw evenly.

The slant of the tool. Diagram (5). This shows the front elevation of the tool, illustrating the clearance between both sides of the tool and the thread faces. It is readily seen that the inclination or slant of the tool is greater than the inclination of the top and bottom of the thread. Therefore the side clearances of the tool must be sufficient to just clear the root angle. It is advisable not to grind any more metal from the tool than is necessary to clear, because the tool requires strength and rigidity.


P
$\frac{1}{2}$P
S
D.
1
P=Pitch = 1 / no. of thds. per inch.
D=Depth= P x .500+clearance
S=Space= P x .500.
Clearance .005"
2
Diameter Standard.
Root Dia.
Bore.
Dia. at bottom of Nut.
Screw
Nut.
3
Screw blank.
Rule.
Tool.
4
Bore.
Rule.
Inside tool.
Pitch.
5
Root Dia.
Tool.
6
A.
Template.
A.
Tool.
7
B.
10°
A.
Pitch.
(Circumference at Root of Thread.)
SQUARE THREAD

Templates for grinding side clearance of the tool. Diagram (6). It is well to use a template when grinding the side clearance. Many beginners have broken square threading tools because the leading clearance angle was only sufficient to clear at the top of the thread, and when the root was approached, the tool was snapped off because of insufficient clearance.

To obtain the angle (A), draw a right-angled triangle, as shown below diagram (6), with the side opposite the angle (A), equal to the circumference at the root of the thread and the side adjacent angle (A) equal to the pitch in a single thread and the lead in a multiple thread. When the hypotenuse of the triangle is drawn, the angle (A) is obtained. Cut out metal templates as shown in diagram (6) and test tool for clearance as shown, while it rests on a metal block. *Example.* To find slant of following and leading side of tool template. If the screw in diagram (5) has $1\frac{1}{4}''$ outside diameter and 4 threads per inch. Lead = .250″, Outside diameter = 1.250″, Root diameter = 1.000″. Circumference (outside) = 3.14 x 1.250″ = 3.92″ (following side); circumference (root) = 3.14 x 1.000″ = 3.14″ (leading side). *Method* 1. Lay these out on metal, cut out and test angles on tool. *Method* 2 by trigonometry.

$$\text{Tan} = \frac{\text{side opposite}}{\text{side adjacent}} = \frac{3.92''}{.250''} = 15.68. \quad 15.68 = \text{tan angle } 86^\circ\ 21' \text{ template angle for following side.}$$

$$\text{Tan} = \frac{\text{side opposite}}{\text{side adjacent}} = \frac{3.14''}{.250''} = 12.56. \quad 12.56 = \text{tan angle } 85^\circ\ 27' \text{ template angle for leading side.}$$

Shape of the tool, Diagram (7), shows the plan and elevation of a tool used for cutting a square thread. The front clearance is 10°, which is sufficient and gives good support to the cutting edge. The front rake is shown lipped to roll the chips clear of the work, and to make the tool cut freely.

CUTTING AN ACME THREAD

The acme thread is used quite often in place of a square thread. Lathe lead screws used to have a square thread but now it is quite common to see an acme thread used. The chief reason for this is that the half-nuts can engage readily with a 29° angle thread without doing damage to the half-nuts or lead screws. The acme thread is also easier to cut than a square thread and is much stronger.

Comparison between U.S.S., acme and square thread. Diagrams (1), (2) and (3). The section of the three threads shown in the diagrams show a comparison of size and shape for threads of equal pitch. In order of root strength, it can be seen that U.S.S. comes first, acme second, and square thread third.

In order of amount of wearing surfaces, a similar sequence will be noticed. In order of the reaction forces from the face of the thread tending to burst the nut, the square thread has no such action, the acme has some, and the U.S.S. a great deal.

It can be seen from the foregoing that the acme thread is a compromise between a U.S.S. and a square thread.

Section of acme screw and nut. Diagram (4). Here one can see that clearance is provided at the top and bottom of the thread, similar to a square thread. The flat (C) at the bottom of the thread will be less than the flat at the top of the thread (F), because the depth equals one half the pitch, plus the clearance.

The bore of the nut is found by subtracting the pitch (P) from the outside diameter of the screw. *The diameter at the bottom of thread of the nut* is found by adding .020″ to the outside diameter of the screw. (*The root diameter of the screw* equals the outside diameter minus the pitch plus .020″ clearance.)

Grinding and setting the acme thread tool. Diagram (5). The profile of the tool is ground to an inclusive angle of 29° and is tested by the angle shown at (A) on the 29° screw thread gauge. The flat at the end of the tool on the top face is measured as shown at (B). When setting the tool at right angles to the axis of the work, the leading edge of the tool must fit snugly against the angular edge on the gauge, while the edge of the gauge fits tightly against the work as shown at (C). When in position to cut the acme thread the front clearance of the tool should be 15° as shown at (D).

Setting the tool for cutting an internal acme thread. Diagram (6). After the profile angle of the inside threading tool has been tested, the

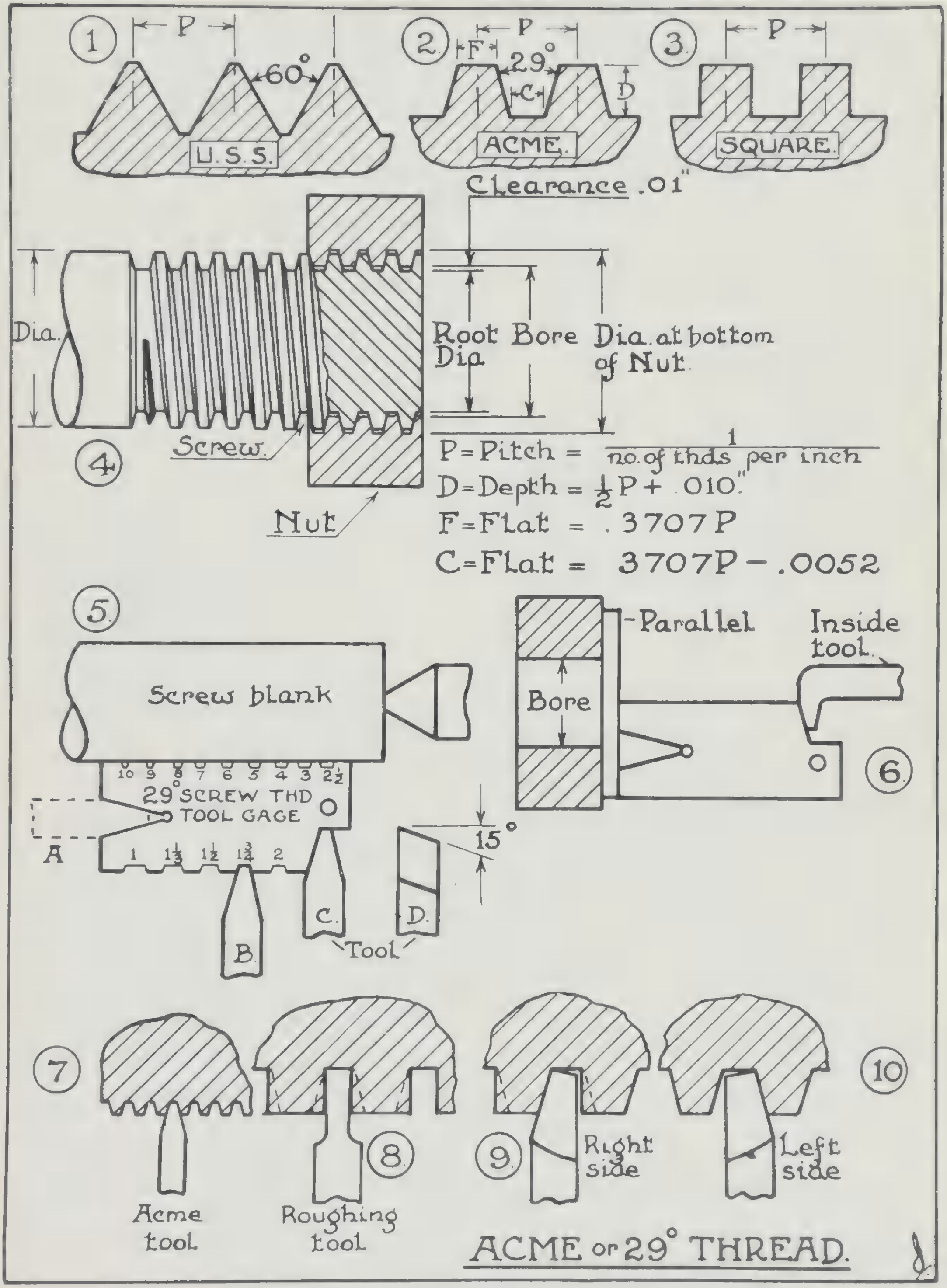
tool is set in position for threading by placing the gauge against a parallel resting across the face of the nut and then setting the tool to the gauge.

Cutting acme threads with fine pitches. Diagram (7). For pitches finer than 5 threads per inch, the same tool may be used for rough and finish threading. This contact on both sides and the front of the tool tends to draw the tool into the work and as in cutting a U.S.S. thread (page 51), precaution as to the type of toolholder used and the height of the tool must be attended to if a smooth thread is to be produced.

Cutting acme threads with coarse pitches. Diagrams (8), (9), and (10), It is better to cut a coarse thread in stages. First, cut a square groove with a tool similar to a square threading tool, but slightly smaller at the end than the finish flat required at the bottom of the thread. This operation is followed by a right side tool, as shown in diagram (9); then followed in the next operation by a left side tool, as shown in diagram (10). The advantage of using these two tools lies in the fact that they cut freely. One ribbon-like chip is removed from each side of the thread at one time and there is no binding action on the tool, so that a smooth face is produced. If desired, a finishing tool may be used the exact size and shape of the thread, with a very light cut.

Table of Acme Standard Threads

Diameter	Threads per inch	Diameter	Threads per inch
1/2″	10	1″	6
5/8″	9	1¼″	5
3/4″	8	1½″	4
7/8″	7	2″	3


1
P
60°
U.S.S.
2
P
F
29°
C
D
ACME.
3
P
SQUARE.
Clearance .01"
Dia.
Root Dia
Bore
Dia. at bottom of Nut.
Screw.
4
Nut
P=Pitch = 1 / no. of thds per inch
D=Depth = ½P + .010"
F=Flat = .3707P
C=Flat = .3707P − .0052
5
Screw blank
10 9 8 7 6 5 4 3 2½
29° SCREW THD TOOL GAGE
A
1 1¼ 1½ 1¾ 2
B.
C.
D.
Tool
15°
Parallel
Bore
Inside tool.
6
7
Acme tool
Roughing tool
8
9
Right side
Left side
10
ACME or 29° THREAD.

CUTTING MULTIPLE THREADS

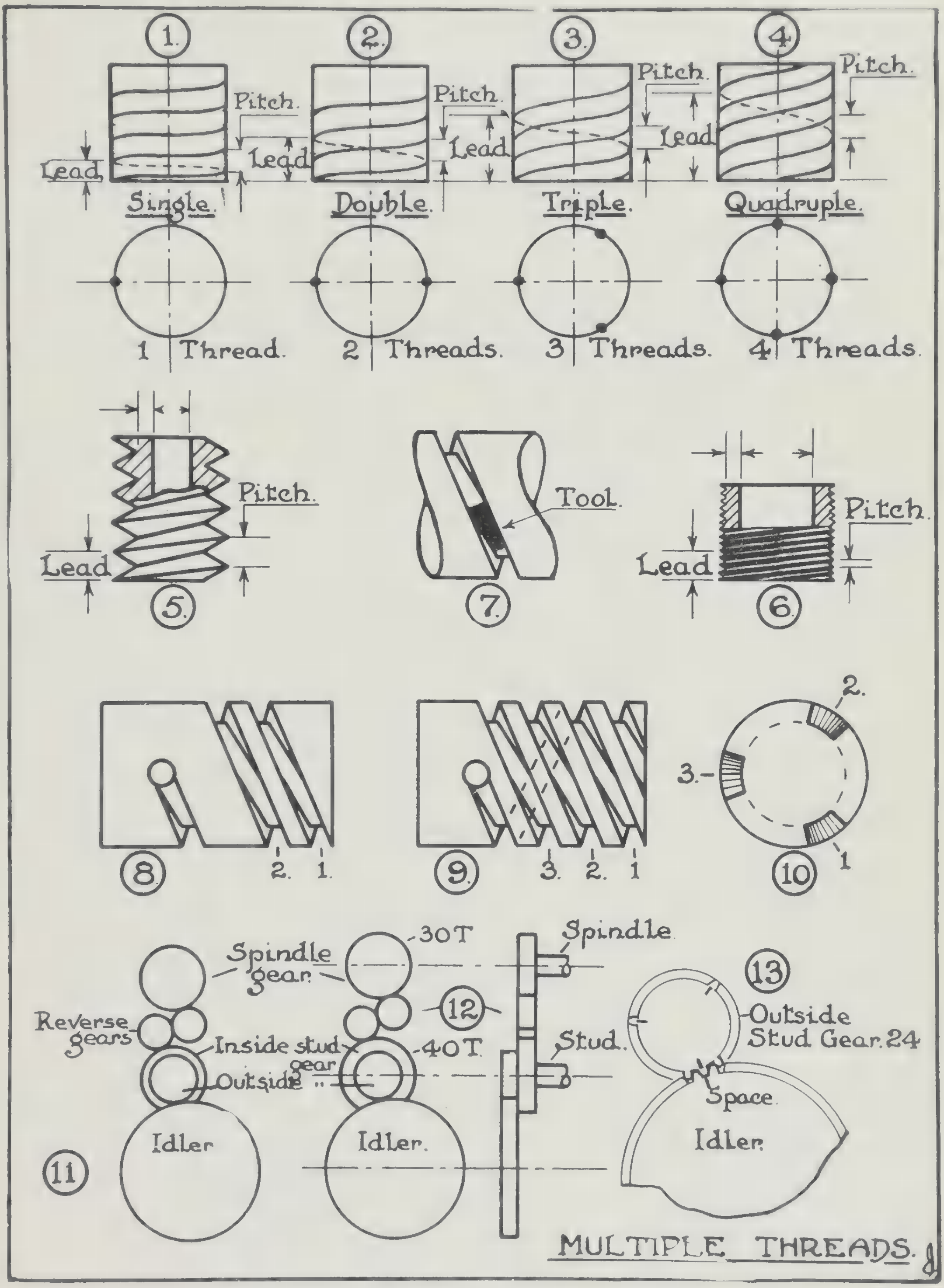
If a screw has multiple threads, it means that it has many threads or more than one thread. Multiple threads are used on screws when a fast lead or great progression is required per revolution of the screw. A common fountain pen is a good example of the necessity for multiple threads. Here it is necessary for the cap to go on a sufficient distance to hold it, without too much turning, and yet the threads must be fine. A quadruple thread is commonly used for this purpose.

Lead and pitch of threads. Diagrams (1) to (4). Diagram (1) shows *a single thread,* or one path or groove passing round the cylinder. In a single thread the "Lead" and "Pitch" are equal. "Pitch" equals the distance from one thread to the next thread. "Lead" equals the amount of progression a screw would make in a nut per revolution of the screw.

Diagram (2) shows a double thread. The end view of the cylinder shows how two threads start at points diametrically opposite each other, the lead in each case being twice the pitch. *Diagram* (3) *shows a triple thread.* The end view shows that three threads start at three points equidistant from one another, the lead in each case being three times the pitch. *Diagram* (4) *shows a quadruple thread.* The end view shows that four threads start at four points equidistant from one another, the lead in each case being four times the pitch.

Comparison between threads of equal lead but unequal pitch. Diagrams (5) and (6). The two screws shown have the same outside diameter, one having a single thread (diagram 5) and the other having a quadruple thread (diagram 6). The single thread is designed to carry a heavy load and the quadruple thread a light load, but both give the same progression per revolution of the screw. Both threads have the same amount of metal from the root of the thread to the bore, and yet it can be seen that the bore in diagram (6) is much greater, which shows that a quadruple thread gives a greater comparative lead with a small wall of metal.

Tool design for a triple thread. Diagram (7). The lesson on cutting a square thread, page 65, shows the importance of the slant of the tool. This slant is further accentuated in the case of multiple threads, as can be seen from diagram (7). In addition to this side slant, it is advisable to slant the top face of the tool so that both sides of the thread are cut with a tool with equal rake.



1.
2.
3.
4.
Pitch.
Lead
Single.
Double.
Triple.
Quadruple.
1 Thread.
2 Threads.
3 Threads.
4 Threads.
Pitch.
Lead
5.
Tool.
7.
Pitch.
Lead
6.
2. 1.
8.
3. 2. 1
9.
2.
3.
1
10
Spindle gear.
30T
Spindle.
Reverse gears
Inside stud gear
Outside "
40T.
12
Stud.
13
Outside Stud Gear. 24
Space
Idler
Idler.
Idler
11
MULTIPLE THREADS.

Cutting a triple square thread. Diagrams (8) and (9) and (10). There are three distinct grooves to be cut in the metal, as shown in the end view, diagram (10). Drill one hole to terminate cut one, and gear up machine to give the required lead. See lessons on gear ratio, page 133. Engage the half-nuts but do not cut the work, and see that the tool terminates in the centre of the drilled hole. If not, offset the tool with the compound slide rest. When one thread is completely cut, it is necessary to index the tool for the next cut. Diagram (8) shows two grooves completed with the third one to be cut; and diagram (9) shows the three threads finished.

Indexing the work to cut a multiple thread. The simplest method of doing this is to use a drive plate on the spindle with slots cut accurately to place the tail of the lathe dog in. For a double thread two slots diametrically opposite would be used. For a triple thread, three slots would be used, and for a quadruple thread, four slots.

Indexing with the stud gear and idler. Diagrams (11) and (12) show that on some lathes the spindle and stud ratio is one to one; that is, the number of teeth in the spindle pinion and inside stud gear are equal, diagram (11). In Diagram (12), there are 30 teeth on the spindle pinion and 40 teeth on the inside stud gear, so that the ratio would be three to four. This must be watched in figuring the gear ratios and in indexing. Diagram (13) shows the method of indexing for a triple thread with spindle and inside stud gear ratio one to one. The outside stud gear should have a number of teeth which is a multiple of three. Mark every eighth tooth with a piece of chalk to give three divisions in a 24 tooth gear. Now mark a space in the idler gear opposite one marked tooth in the stud gear, while the first thread is being cut. To index for the second thread, disconnect the idler and turn the spindle until the next tooth on the stud gear engages the space in the idler gear; but do not turn the idler, or the lead screw position will be incorrect. Now cut the second thread. Then proceed similarly with the third thread.

LATHEWORK QUESTIONS

1. Name the important mechanisms to be found in an apron mechanism.
2. Sketch the gear mechanism of a lathe apron when engaged for: (a)automatic cross feed, and (b) automatic long feed.
3. Sketch the mechanism used when operating by hand a long feed.
4. Why is it necessary to prevent the half nuts and the friction feed being engaged at the same time?
5. Which threads are best for a lead screw of a lathe (a) Square threads (b) Acme threads?
6. Which is preferable in a lathe: (a) One that has a lead screw splined for turning the worm in the feed mechanism; (b) one that has a lead screw and a separate splined shaft for automatic feed? Why?
7. Sketch the half nuts of a lathe in two views each (a) half nuts engaged (b) half nuts disengaged.
8. Sketch a piece of work rough formed to a profile.
9. Sketch a forming tool that contains a complete form on its cutting edge.
10. Which is better, to finish up on a formed shape or finish down? State the reasons why.
11. Sketch a spring forming tool. What is the advantage of using such a tool?
12. How can a template be traced from a blue-print of a formed part? Illustrate by a sketch.
13. What are the advantages, if any, of the use of a hand tool when forming?
14. What assistance can the automatic feeds of a lathe give when forming?
15. Sketch a 1″ mandrel showing all its features with the centre hole in section.
16. Sketch a mandrel showing the taper, the small end, the large end and the location of the nominal diameter.
17. Why are flats ground on a mandrel?
18. Why is a mandrel made of tool steel and why is it hardened, tempered and ground?
19. State two methods of putting work on a mandrel.
20. Sketch a mandrel with work mounted on it and show the large end in relation to the direction of the feed of the tool.
21. How would you prevent a pulley from slipping on a mandrel while being turned?
22. Why should work be reamed before mounting on a mandrel?
23. Which is preferable when placing work on a mandrel, (state reasons why (a) A soft hammer, or (b) an arbor press?
24. Why is it necessary to "spot" work before drilling it in the lathe?

25. Sketch in section a "spotted" hole showing a drill point starting to drill. What should be the relation of the angle of the spotted hole compared to the angle of a drill point?
26. Sketch two tools commonly used for spotting, previous to drilling.
27. What are the disadvantages of placing taper shank drills in a lathe tailstock for drilling work?
28. Sketch a drill holder for use on a lathe. What are its advantages?
29. Why is it necessary to ream a hole after drilling? What reamer should be used on a lathe? Where does the cutting take place on the reamer?
30. What sized drill should be used to drill a hole that has to be reamed with a ¾″ reamer?
31. Why is it necessary to have drills ground accurately when drilling holes previous to reaming?
32. Why should the work being reamed not be reversed when backing out a reamer? What happens if the reamer is taken out of the hole without turning the work in the lathe?
33. How would you locate the centre of round stock?
34. How would you lay out the centre lines on the end of round stock so that both lines would be in the same plane?
35. How would you check the "offset" of a piece of work which is centre dotted only on the ends, when making an eccentric shaft?
36. By what method is the offset centre of a piece of stock drilled when making an eccentric shaft?
37. Sketch two end views of two centre holes for an eccentric shaft: (a) Correctly drilled, (b) Incorrectly drilled.
38. How can centre holes be corrected if the "offset" is slightly out?
39. Make a sketch showing the "offset" and the "throw" of an eccentric shaft.
40. Sketch an eccentric shaft being turned on centres showing the type of tool used to reduce the eccentric ends.
41. Sketch three positions of the reverse gears of a lathe showing Reverse, Forward and Neutral positions.
42. Sketch the spindle gear, reverse gears and stud gear so that the stud gear makes ¾ of a revolution each complete revolution of the spindle.
43. Sketch the front and end views of a lathe gearing which is set with a simple train of gears to cut a right hand thread.
44. Sketch the front and end views of a lathe gearing which is set with a compound train of gears to cut a left hand thread.
45. How do you decide to set up a simple or a compound train of gears?
46. What gears are not changed and what gears are changed when setting up lathe gears?

47. What is likely to occur when cutting occurs on both faces of a U.S.S. tool when cutting a thread?
48. Sketch the view of a U.S.S. thread tool cutting a thread, showing the height of the tool with regard to the axis of the work.
49. What is the influence of too much:—(a) front rake, (b) front clearance, (c) side clearance, when cutting a U.S.S. thread?
50. Sketch the plan of a U.S.S. thread tool showing the direction and angle of the cut when using the compound slide rest to feed the tool into the work.
51. Why should free cuts be taken occasionally when cutting a U.S.S. thread?
52. What should be the depth of each cut when cutting a U.S.S. thread: (a) When beginning, (b) When finishing?
53. Why should a thread gauge be held horizontally when setting a thread tool?
54. Why is it necessary to set the thread tool in the correct position with regard to the axis of the work being threaded?
55. Sketch the plan of a thread tool being tested for position, showing the thread gauge and the work to be threaded.
56. How is a thread tool set when cutting a thread on tapered work? Is it set with the thread gauge parallel to the tapered face of the work or parallel to the axis of the work?
57. Sketch a spring threading tool. What are the advantages of using such a tool when cutting a thread?
58. Sketch one type of threading tool which provides the profile of the tool ground ready for use.
59. Sketch the plan of a threading tool cutting to a shoulder, on work with thread clearance provided by means of a groove cut near the shoulder.
60. Name two methods of testing the number of threads per inch on work being threaded in the lathe.
61. Sketch a caliper suitable for testing the outside diameter of threaded work.
62. Sketch the anvils of a thread micrometer testing threaded work.
63. What diameter does a thread micrometer measure?
64. What would be the measurement of a thread micrometer when measuring a 1″—8 U.S.S. thread?
65. Sketch the micrometer anvils measuring a U.S.S. thread by the three wire method.
66. What is the formula for measuring the diameter of threaded work by the three wire method?
67. Sketch a thread dial mechanism showing the dial, the worm gear and the lead screw of a lathe.

68. If a thread dial has eight main divisions marked on it, and the worm gear has thirty-two teeth meshing with a lathe lead screw, what amount of travel of the carriage does each division of the dial indicate?

69. What divisions on the thread dial, as described in question (68), can be used for correctly indicating when to connect the half nuts of a lathe with a four pitch lead screw? (a) For four threads per inch, (b) For nine threads per inch, (c) For $4\frac{1}{2}$ threads per inch, (d) For $11\frac{1}{2}$ threads per inch.

70. What do you understand by an even geared lathe?

71. If the lathe dial as described in question (68) had eight main divisions and sixteen half divisions, how could the half divisions be used to advantage?

72. Sketch an enlarged section of an external square thread showing the proportions of the thread.

73. Sketch a section through a square thread and nut.

74. What size would you bore a nut to be square threaded to fit a $1\frac{1}{4}''$ diameter screw, four threads per inch?

75. Sketch three views of a square threading tool.

76. Calculate the angle of a template for testing the angle of the leading and following sides of a square threading tool. The thread to be cut is $1\frac{1}{4}''$ diameter and four threads per inch.

77. If a square thread tool is not ground with sufficient side clearance what is likely to happen when cutting the thread?

78. Show by means of sketches how a square thread tool is set correctly in relation to the work for:— (a) An external thread, (b) An internal thread.

PLANING IN THE SHAPER

THE SHAPER MECHANISM

After the student is familiar with the lessons in book one, and has operated the shaper on simple operations, it would be well to investigate further the mechanism of the shaper. The shaper offers some some interesting examples of applied mechanics. Primarily, it is an example of a machine working at a mechanical disadvantage because the driving action of the crank pin on the vibrating arm acts as the third order of lever.

Position of stroke. Diagram (1) and (2). If the vibrating arm is fastened to the ram in the position shown in diagram (1), the stroke at the tool point will occur near the column of the machine. If the vibrating arm is fastened to the ram at the rear of it, as shown in diagram (2), the stroke at the tool point will occur farther away from the column than that shown in diagram (1). Work should be so placed that the stroke will cover it nearest to the column for greater support and rigidity.

Mechanical diagram of shaper with fixed fulcrum. Diagram (3A). It can be seen from the line diagram that the forward or cutting stroke takes up far more rotation of the crank pin than the return or non-cutting stroke. Since the crank pin rotates at an even rate, it must follow that the return stroke is much quicker than the forward stroke. This is the method of producing in the shaper the *Quick Return* during the non-cutting stroke. The power or lever arm during the cutting stroke is longer than during the return stroke, giving greater power when it is needed during the cutting stroke. With a fixed fulcrum at the bottom of the vibrating arm, the top of the arm is compelled to move on the arc of a circle. To allow for this arc-like movement when fastening the vibrating arm to the adjusting screw, the block, as shown in diagrams (3B) and (3C), is slotted, allowing the end of the arm to move up and down the slot while passing through the arc.

Detail of crank pin and vibrating arm. Diagram (4A). This diagram shows the crank pin with its location on the face of the bull gear, which controls the length of stroke of the machine. The crank pin rotates in the block which slides up and down the slot in the vibrating arm, forcing the vibrating arm to move. This vibrating arm has a fixed fulcrum at the bottom; therefore the top end moves on the arc, as previously illustrated in diagram (3A). Diagram (4B) shows another method of connecting the vibrating arm to the adjusting screw. Here a

link is used to connect the end of the arm to the block through which the adjusting screw passes.

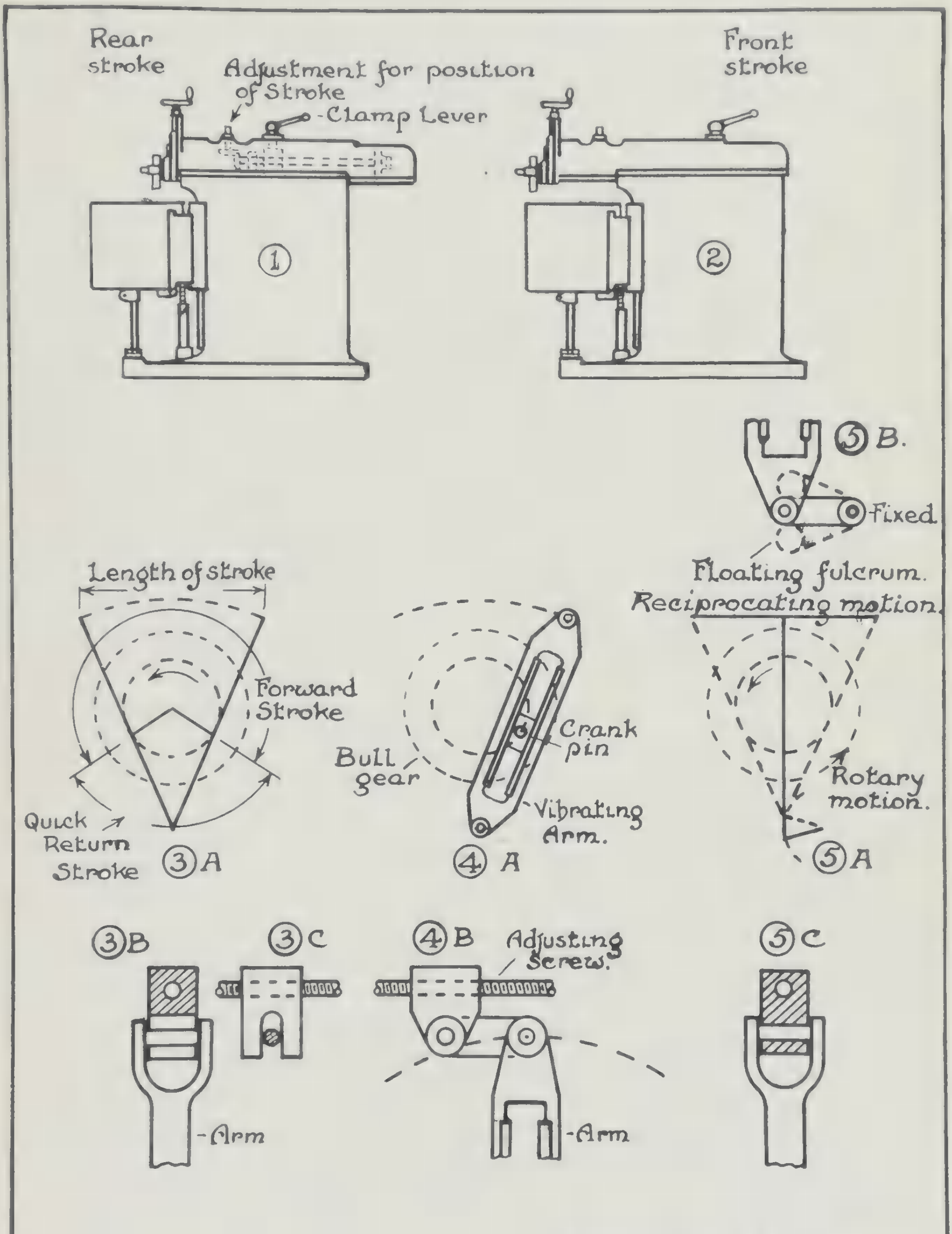
Vibrating arm with floating fulcrum. Diagrams (5A) and (5B). The bottom end of the vibrating arm is connected to the fixed fulcrum through a link, as shown at (5B). As the crank pin rotates and moves the arm, the bottom end of the arm moves up and down on the floating fulcrum because the top end is fastened to the adjusting screw block (5C) and must move in a straight line with it in a reciprocating motion.

Cutting speed of the shaper tool. In Diagram (3A), the return stroke arc represents the time of the return stroke, while the forward stroke arc represents the time of the forward stroke. The relation of these arcs is usually about 2: 3 or 1: 1½. If, therefore, the bull gear makes 150 RPM., and the stroke of the machine is 12″ long, the tool goes forward and back 150 times in one minute, that is, the tool actually covers 300 feet per minute, but the cutting stroke takes 3/5 of the time and the return stroke 2/5 of the time.

Therefore the cutting speed equals 5/3 x 150 feet or 250 feet per minute and the return stroke of the tool equals 5/2 x 150 feet or 375 feet per minute. The length of the stroke governs the cutting speed, not the length of the work being cut, because for a given rotating speed of the bull gear, the ram moves a distance according to the position of the crank pin.

If the work is 4″ long and the stroke 6″ long, the cutting speed must be figured on a 6″ stroke for a certain RPM. of the bull gear.

To obtain the same cutting speed in FPM (feet per minute) for a 6″ block as for a 3″ block, speed changes must be made either by gears in the machine or steps on the cone pulley.



SHAPER MECHANISM.

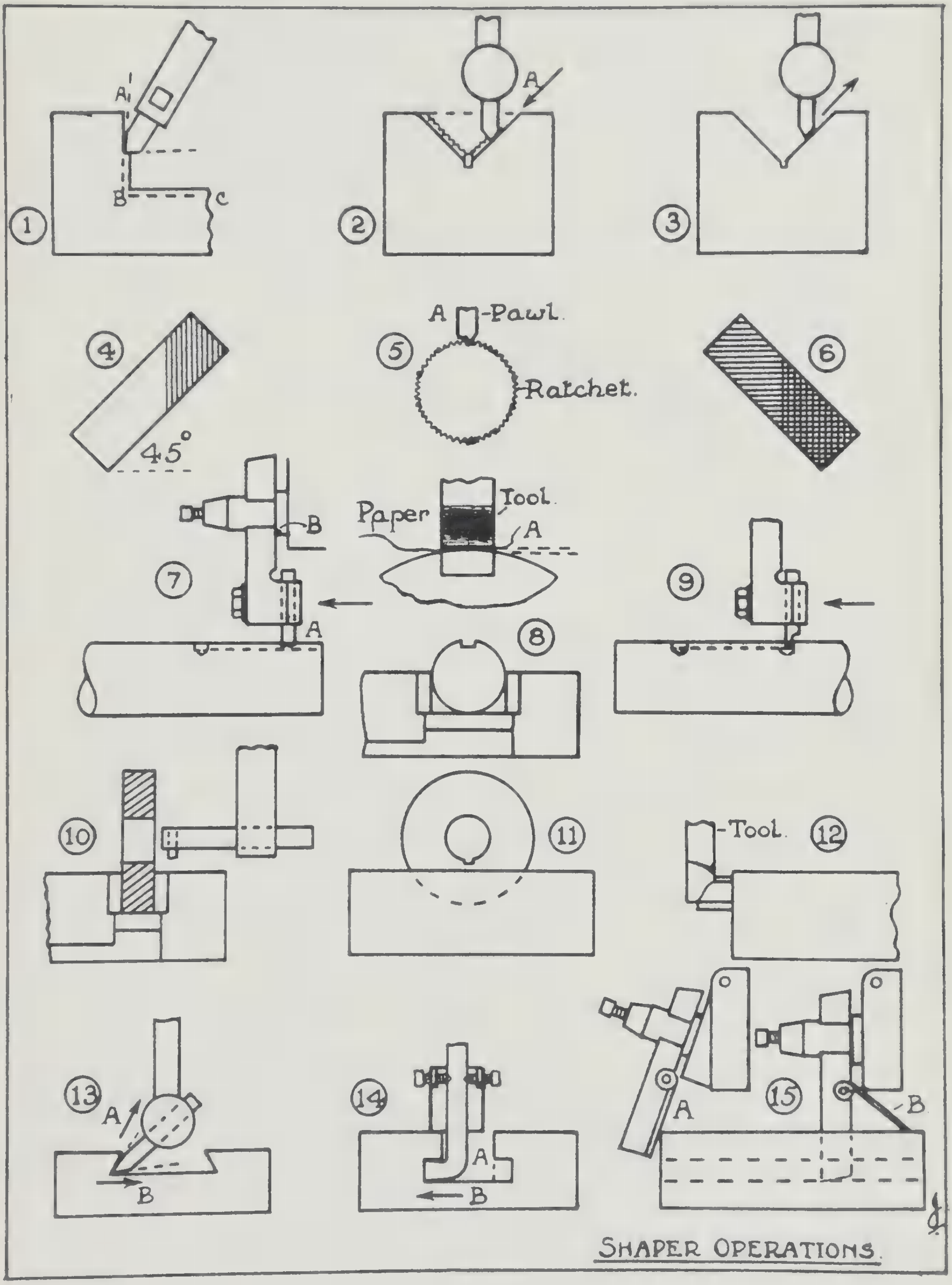
SHAPER OPERATIONS

Planing a right-angled corner. Diagram (1). This type of shaping is well illustrated in the making of a toolmaker's vise where there are two right angle corners at each end. The tool used for this operation is ground to a profile angle of 80° to give clearance on both sides of a right-angled corner. The swivel head is set at zero and for the down feed, the rough cut is made with the horizontal edge of the tool cutting. The finish cut is made when the tool is fed upwards. It is very important that the apron swivel be swung to the right for the cut (AB), so that the tool lifts away from the vertical face on the return stroke. The horizontal face (BC) is rough machined in the direction (C) to (B) with the vertical face of the tool, then finish planed in the direction (B) to (C) with the horizontal face of the tool.

Angular planing. Diagram (2). This is illustrated in a typical manner as when shaping a Vee block with an angle of 90°. If the Vee is cut from the solid metal, a slot is first cut with a parting tool after which the stock is roughed out using the horizontal feed. The apron swivel is set at 45° for the angular cut and the apron swivel is swung away from the face, being cut so that the tool will not lift into the face on the return stroke. The face is rough cut down to the angle as shown at (A), diagram (2), then the finish cut is made on the up stroke, as in diagram (3), after which the opposite side of the angle is similarly planed.

Indexing and cutting serrations. Diagrams (4), (5) and (6). This operation is illustrated by serrating the steel jaws of a vise. The work is held in the shaper vise and the vise is swivelled to 45°. The cut of the tool is set to a suitable depth and the horizontal feed is controlled by the movement of the ratchet, which is in turn moved by the pawl (A), the amount of the feed being controlled by the off-set on the rocking lever (see book 1, pages 88 and 89). If the feed screw has 4 threads per inch and is a single thread and the ratchet has 50 teeth, a one-tooth feed would move the work .250″/50=.005″. If the pitch of the serrations were required to be .030″, then the pawl must engage every sixth tooth. Start the machine and the pawl will index the serrations automatically. When one angular cut is made all over the work, swing the vise at 45° in the opposite direction and repeat the operation as in diagram (6).

Cutting a keyway on the end of a shaft. Diagram (7). The shaft is shown held in the vise diagram (8). A hole is previously drilled in the shaft in which the cut will finish. A tool is ground the correct width of the keyway and the stroke is set to terminate at the centre of the hole.


A₁
B
C
A
A
Pawl.
Ratchet.
45°
Paper
Tool.
A
B
A
Tool.
A
B
A
B
A
B
SHAPER OPERATIONS.

A smooth cut will be made if the tool bit is placed at the back of the holder at A, diagram (7), when the cutting edge is behind the last point of support B. Set the tool with paper as in enlarged diagram (8A). The depth of the cut will equal the finish depth of the sides of the keyway plus the depth of the arc of the shaft. This amount can be measured with the graduated dial and can be found by looking up the table in a data book. The keyway can also be cut direct and the depth at the sides of the keyway measured by trial.

Cutting a keyway in the middle of a shaft. Diagram (9). The keyway has to be cut between two holes previously drilled in the shaft. The stroke of the machine must be set so that the front edge of the tool terminates on the centres of both holes. It will be necessary to grind clearance at the back of the tool, as shown, to prevent it striking against the shaft. If the clearance is ground on the angle, the tendency will be for the tool to lift easily if it touches the shaft on the return stroke.

Cutting a keyway in a gear. Diagrams (10) and (11). The gear is held in the vise as shown and the tool bit is fastened in a bar held in a special tool holder. The bar should only extend from the holder a sufficient distance to clear the work when the stroke is set. The depth of the keyseat is measured by the graduated dial on the down feed from the point at which the tool bit first contacts with the work.

Forming the edge of a rectangular block. Diagram (12). The forming tool is ground to suit the profile of the work, which is held protruding from the shaper vise. Smooth work will be produced if the tool bit is held at the back of a special spring planer tool similar to that shown in diagram (7).

Planing a dovetail. Diagram (13). If the dovetail has an angle of 60°, the swivel head will be set at that angle and the tool bit ground at 50° for clearance. The apron swivel must be swung away from the face of the dovetail. The finish cut is made in the direction of the arrows A and B.

Planing a tee slot. Diagrams (14) and (15). It is absolutely necessary in this operation to fit some form of a hinged lifter which will slide over the top surface of the work on the cutting stroke as at (B) diagram (15) and lift the tool entirely out of the slot (A), diagram (15) on the return stroke. The tee slot is first grooved as at (A) dotted diagram (14), then the teeslotting tool is fed into the work in the direction of the arrow (B) diagram (14).

SHAPER TOOLS

Shaper tools include various types of tools, some forged from solid steel or tool bits ground to various shapes and used in various types of toolholders. In addition to these, one should be familiar with the various methods of holding down work on the shaper while it is being planed. The proper setting of the toolholder while cutting is important if good work is to be produced.

Position of the toolholder. Diagram (1). The toolholder should be placed, if at all possible, in a position so that the cutting edge is not directly beneath the pivot point of the toolholder which is at the centre of the toolpost (A). It must be inclined slightly away from the direction of the feed of the work (B) so that, if it turns under undue pressure, it can only turn away from the work, making it larger. If it were so placed that it inclined towards the direction of the feed, it might be turned into the work and make it too small. Due to this tendency of the tool to spring away from the work, chatter marks will be often avoided and a nice finish assured.

The finish planing. Diagram (2). The tool bit will be placed at the back of the spring toolholder and the holder inclined as in diagram (1). When rough planing as shown in diagram (1), it is advisable to swing the apron swivel (C) slightly away from the direction of the feed so that on the return stroke the tool will lift away from the metal.

Planing an undercut. Diagram (3). If the surface (AB) cannot be cut in any other but a horizontal position, it will be necessary to arrange the tool bit and toolholder as shown. If a cut were made in this position, as the tool returns, the clapper box would lift as the tool contacts at AB. If the tool lifted on the return stroke, it would of necessity lift into the work and damage it. In this case, therefore, it is necessary to lock the clapper box to prevent it from lifting.

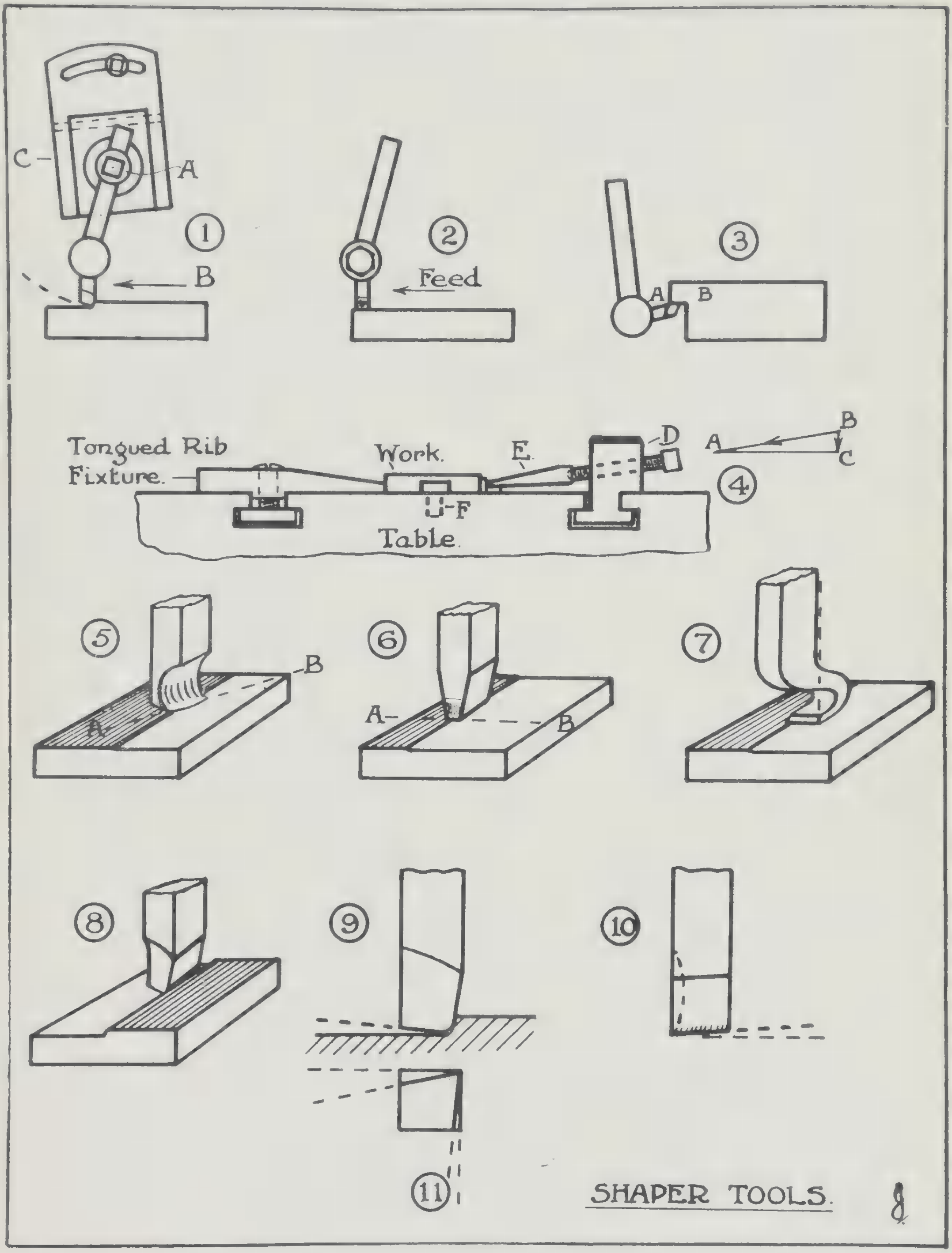
Holding down work to the table. Diagram 4. Thin work similar to that shown is held down by the pressure of a screw bunter (D), which fits into the tee slot of the table. The screw is at an angle of 10° and presses against the finger or toe dog (E). The force (AB) acting at an angle has a vertical component (BC) which presses the work down to the table. In addition to this pressure on the work, a stop pin (F) fits into a hole in the table and acts as a stop on the end of the work. One can readily appreciate that both in holding down work and in grinding and putting the tool in the correct position, the forces acting upon the tool have a great deal to do with the successful operation of the shaper.

Finish planing steel. Diagram (5). The shear tool illustrated will produce a smooth mirror-like finish on mild steel or tool steel with a fine feed, fine depth of cut and a cutting compound. The line (AB) shows the shear angle which may be from 45° to 60° to the direction of the feed. The cutting angle of the tool is made slightly convex with about 7° clearance at the back of the cutting edge.

Finish planing with narrow-faced tool. Diagram (6). The cutting edge of this tool, indicated by the line (AB) is slightly inclined to the horizontal and is narrow and straight and produces a nice finish on cast iron with a fine feed and fine depth of cut.

Finish planing with broad-faced goose-neck tool. Diagram (7). This tool would chatter if it were not for the fact that the cutting edge is behind the last point of support and consequently the cutting edge tends to spring away from the metal. It is used with a fine depth of cut and a course feed on cast iron.

General tool for rough planing. Diagrams (8), (9), (10), (11). This tool is shown cutting metal in diagram (8). The profile is shown in diagram (9) with just sufficient clearance to pass the finished surface. There is a good mass of metal behind the cutting edge to support it while cutting and to dissipate the heat. The round nose produces a smooth surface and gives greater strength and wearing quality to the point of the tool. The clearance behind the cutting edge is shown in diagram (10) usually about 7°. Diagram (11) shows the end view of the tool with side clearance and side rake which will vary with the different metals being cut.


C
A
1
B
2
Feed
3
A
B
Tongued Rib
Fixture.
Work.
E.
D
B
A
C
4
F
Table.
5
B
A
6
A
B
7
8
9
10
11
SHAPER TOOLS.

QUESTIONS ON SHAPER WORK

1. Make a freehand sketch using a single line diagram to illustrate the Quick Return mechanism of a Crank Shaper.
2. Illustrate by means of two freehand sketches, the location of the Crank Pin, Bull Gear and Vibrating arm for: (a) A short stroke, (b) A long stroke.
3. Sketch two forms of fastenings for fastening the end of the vibrating arm to the Ram Screw.
4. Sketch the end of a vibrating arm showing a "floating fulcrum." Why is this necessary?
5. Should work be placed near to the column of a shaper, or further away from the column? State why.
6. When installing a shaper why is it necessary to watch that the drive pulley runs in the correct direction?
7. What order of lever is the vibrating arm of a shaper? What advantages and disadvantages does this method of obtaining reciprocation from rotation have?
8. Illustrate by means of a sketch a tool cutting a right angled corner. Show the position and direction of the tool for roughing and finishing: (a) Vertical face of work, (b) Horizontal face of work.
9. Illustrate by three sketches the method of cutting a vee groove for a vee block. Show the three stages of the work.
10. If the swivel is set at 45° for an angular cut, how will the apron swivel be set? Why?
11. If the horizontal feed screw of a shaper has four threads per inch, and the ratchet on it has fifty teeth, how many teeth will the ratchet move at one time to index serrations on the work of .015″ pitch?
12. How would you measure exact to .001″ the depth of a keyway to be cut in a shaft?
13. Sketch the best tool and show it in the best position, for cutting a smooth keyway.
14. How must a tool bit be ground for cutting a sunk keyway between two drilled holes?
15. Sketch the type of tool holder used to cut a keyseat in a gear blank.
16. Why should the length and position of stroke be set before the tool enters the hole of the gear blank being cut as in question (15)?
17. What type of tool should be used when forming work held in a shaper?
18. Sketch a dovetail being cut showing the position of the tool and the direction of the horizontal and angular feeds.
19. How is a tool set up for cutting a tee slot? Sketch the first and second stages of the cutting of a tee slot.

20. Sketch the front view of a toolholder, showing the best position with regard to the tool post and the work, to prevent it from turning into the surface of the work if it strikes a hard spot.
21. Sketch the type of tool you would use for making an undercut. What precautions are necessary for such a cut?
22. Sketch a method of holding down thin work to the shaper table.
23. How is the vertical force obtained in the hold-down illustrated in question (22)?
24. How is a smooth finish obtained when planing mild steel or tool steel? Sketch the type of tool used.
25. What type of toolholder should be used if the tool bit has a broad cutting edge?
26. Sketch three views of a shaper tool point for rough planing cast iron.
27. What kind of surface is produced on cast iron (a) with a sharp point on the tool or (b) with a rounded point on the tool, when used with a fine feed?

MILLING

DIRECT AND PLAIN INDEXING

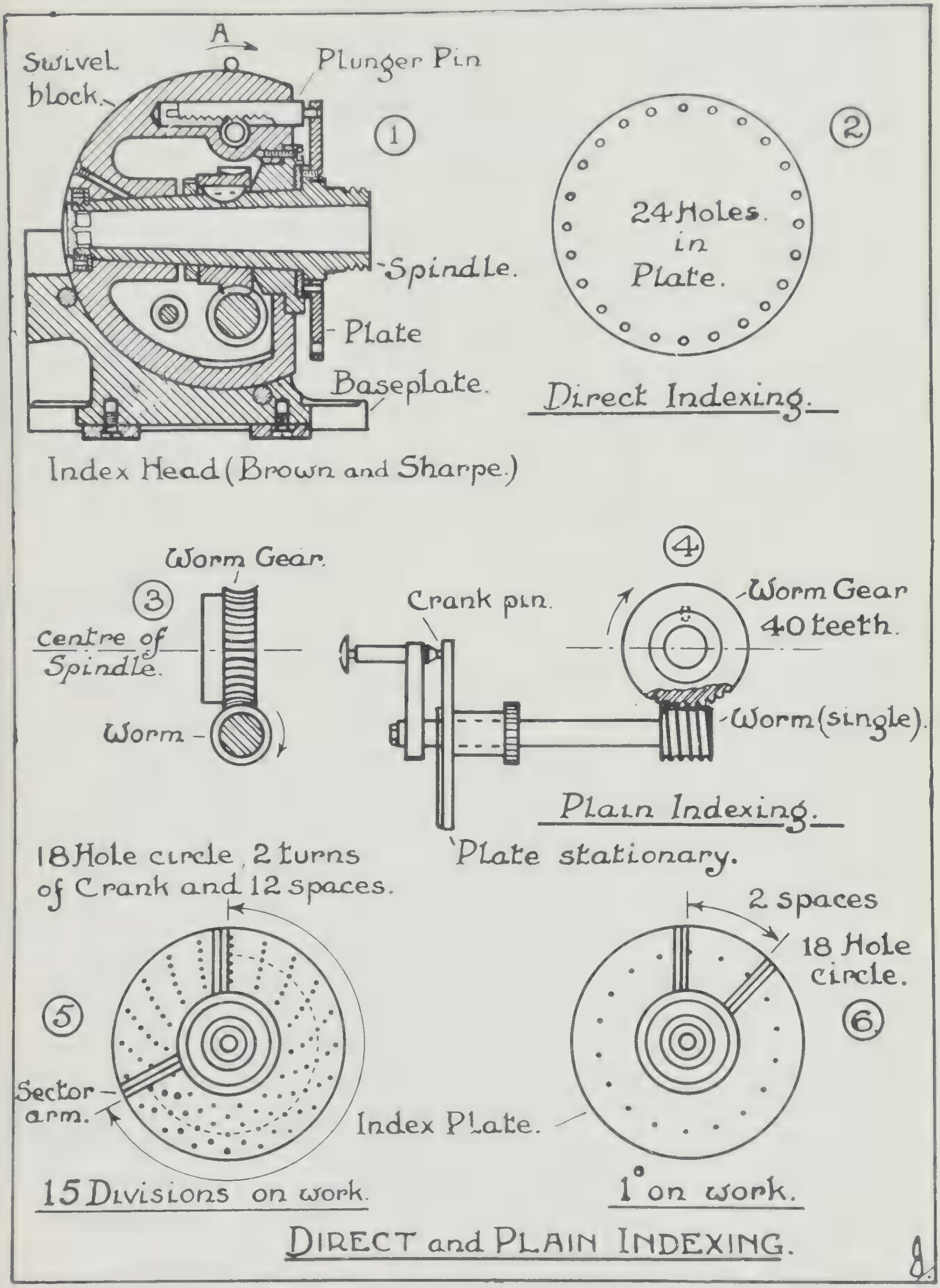
Indexing with an index head offers one of the most important and most interesting operations on a milling machine. It is used for accurately spacing and dividing work, such as is necessary in cutting teeth on a spur gear, and in many other operations. It is a very positive form of measuring device because the measuring is accomplished through very definite movements of a worm and worm gear and the fitting of a pin into holes. This is quite different from measuring with such tools as a rule, vernier, or micrometer, where sight and line-to-line reading is necessary for accurate results.

Section of index head. (Brown and Sharpe). Diagram (1). The index head as a unit fits accurately at its base into the grooves of the tee slots of the milling machine table. This guarantees alignment with the table and the tailstock, which is similarly fitted. The spindle of the head has a Brown and Sharpe tapered hole through it into which fits the live centre, and the nose of the spindle is threaded to receive a chuck or drive plate for the dog which is necessary to turn the work. Against the shoulder of the spindle there is fitted an index plate used for direct indexing. Inside the head a worm gear is keyed to the spindle and this worm gear has 40 teeth and meshes with a single threaded worm, which may be turned to disconnect the worm and worm gear when necessary. The index head may be swivelled so that the axis of the spindle may be set at an angle or in a vertical position.

Direct indexing. Diagrams(1) and (2). This represents the simplest form of indexing, and can only be used for division of 24 or such numbers as 24 is a multiple of. A plunger pin operated by a lever (A) can be made to engage with holes in the index plate and locks the spindle positively when set. It is necessary to have the worm disconnected from the worm gear for this operation to allow the spindle to be moved by hand. *Example.* To mill a hexagon, make one cut, turn spindle 4 spaces, mill next cut, etc. 24/6=4 (count spaces not holes).

Plain indexing. Diagrams (3) and (4) show the necessary parts of the index head used for this operation, the other parts of the head have been left out to simplify the diagrams.

Diagram (3) shows the front view of the worm and worm gear and diagram (4) the side view.


A
Swivel block.
Plunger Pin
1
Spindle.
Plate
Baseplate.
Index Head (Brown and Sharpe.)
2
24 Holes. in Plate.
Direct Indexing.
Worm Gear.
3
Centre of Spindle.
Worm
Crank pin.
4
Worm Gear 40 teeth.
Worm (single).
Plain Indexing.
18 Hole circle, 2 turns of Crank and 12 spaces.
Plate stationary.
2 spaces
18 Hole circle.
5
Sector arm.
Index Plate.
6
15 Divisions on work.
1° on work.
DIRECT and PLAIN INDEXING.
8.

The index plate that the crank pin enters is held stationary while the crank rotates. This turns the worm and worm gear. For one revolution of the crank, the worm gear and index head spindle turns 1/40 of a revolution and this is the basis of all calculation. This ratio of rotation may be used for all divisions that 40 is a multiple of.

A greater range of divisions may be obtained by using in conjunction with the worm and worm gear ratio accurately divided plates with the holes running concentric but varying in number. To keep a check of the number of holes used, two sector arms are set to mark the holes. Diagram (5). As before stated, spaces between the holes should be counted for any given number of divisions.

Example. To divide work into 15 divisions. Diagram (5).

$$\frac{40}{15} = 2\frac{10}{15} = 2\tfrac{2}{3} \text{ turns}$$

Select a circle of holes the number of which is a multiple of the denominator of the fraction. 18 is a multiple of 3. Now multiply the numerator and denominator of the fraction by 6.

$$2\frac{2 \times 6}{3 \times 6} = 2\,\frac{12}{18} \text{ turns.}$$ Set the sector to 12 spaces in an 18 hole circle.

To index for 15 divisions. Turn the crank 2 complete turns and 12 spaces in an 18 hole circle; lock the spindle of the head and make the cut on the work.

To index for degrees. Diagram (6). 40/360=1/9 of a turn. Select an 18 hole circle, multiply numerator and denominator by 2 to make denominator equal to 18; e.g., $\frac{1 \times 2}{9 \times 2} = \frac{2}{18}$ turns.

Turn crank to 2 spaces in an 18 hole circle for 1° division on work.

THE PRINCIPLE OF INDEXING AND FLUTING A REAMER

A reamer has its flutes spaced irregularly to avoid chattering. The teeth are even in number and the lands are placed diametrically opposite so that the diameter can readily be measured with a micrometer. The purpose of this lesson is not to give specifications or tables for indexing reamers but to explain the principle underlying indexing when irregular flutes are required.

Position of cutter in relation to the reamer blank. When a reamer blank is mounted on the centres of a milling machine, it is necessary to place the cutter so that the side that forms the cutting edge of the flute is radiating from the centre of the reamer, as shown in diagram (1). It is necessary to raise the table of the machine until the cutter is at the actual cutting position for depth, then place a small rule against the side of the cutter that forms the cutting edge of the flute, and observe that the rule points exactly towards the centre.

Varied positions for different diameters of reamers. Diagram (2) shows clearly that a reamer blank is offset more for a large reamer than for a small one to satisfy radial conditions.

Experiment (1). If a reamer were indexed as shown in Diagram (3) with all flutes of equal depth, it will be observed that the lands vary in size. This variation is due to the varying number of holes used when indexing and the land sizes are directly proportional to it.

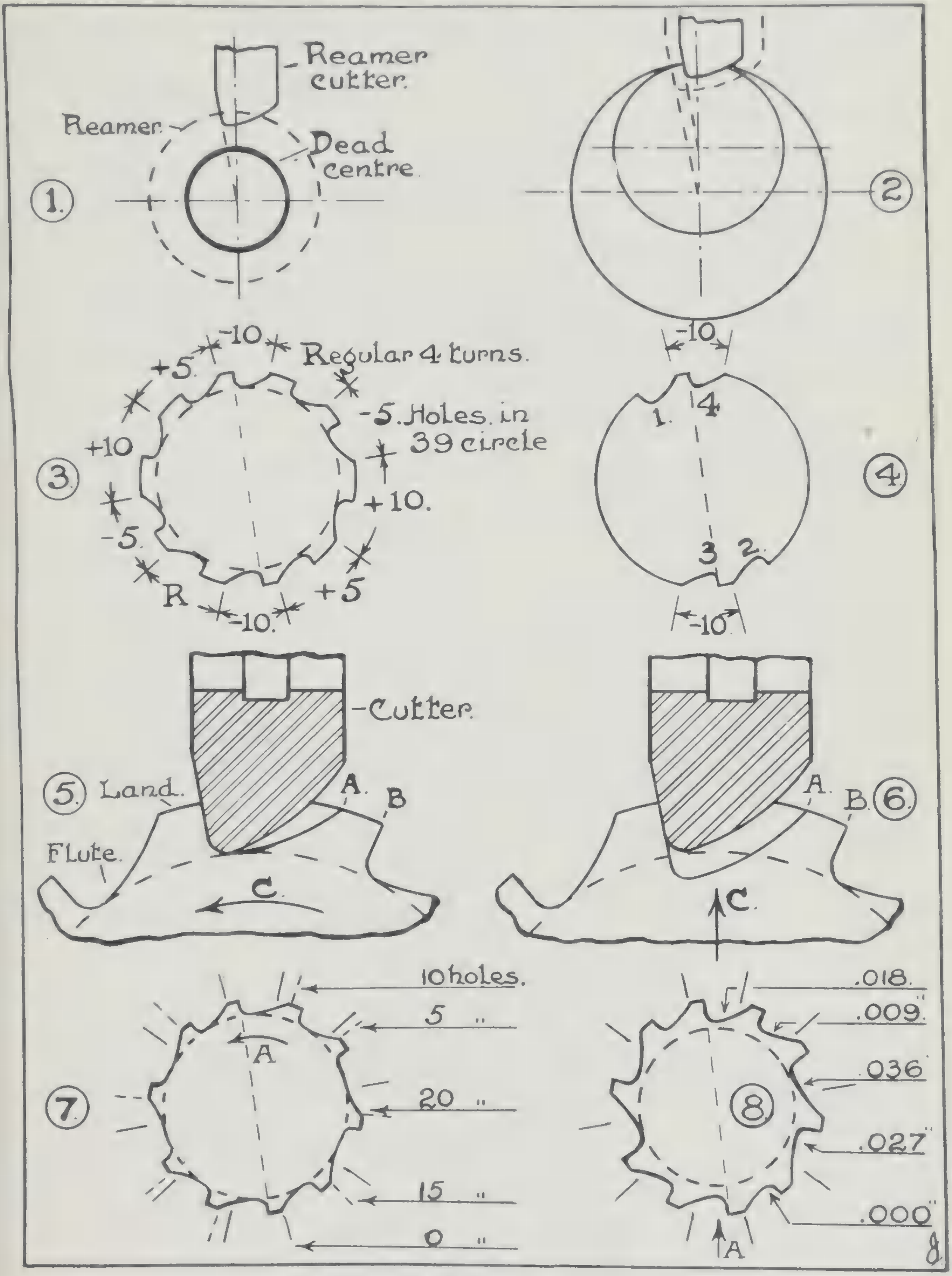
Correction of land sizes. If the lands were made equal by turning the reamer blank around, but still cutting to the same depth as shown in diagram (5), it is obvious that the index or turn made for this correction would be an amount equal to the number of holes given to this spacing beyond that required for the smallest land spacing on the reamer blank. A. B. in diagram (5) shows a corrected land by turning the reamer around the required amount in the direction of the arrow C. If reference is made to diagrams (3) and (7) it will be seen that the index in holes for the correction of the land sizes equal to the small land shown by dotted line, is equal to the original difference in the index from that of the first land established, as shown in diagram (4) where the cuts are made in the order (1), (2), (3), (4).

Regular index turns,—10 holes, gives the small land. Diagrams (3), (4), (7).

Regular 4 turns, take 10 holes off to make it equal to the small land.
Regular 4 turns—5 holes, take 5 holes off.
Regular 4 turns+10 holes, take 20 holes off.
Regular 4 turns+5 holes, take 15 holes off.

Correcting land sizes by increasing depth of cut. Diagrams (6) and (8) show the usual method of cutting irregular flutes to produce equal lands by increasing the depth of cut when the lands are large. The table is raised in the direction of the arrow C. so that the cutter removes the metal required, leaving the land A. B. It can be seen that the depth increase will be of equal ratio to the turn around, or index increment, so that once a depth increase is found correct for the 4 turn—5 hole index, the remainder will vary in the same proportion.

Example. For a 4 turn—5 hole index, the increment depth cut is .009″ more than the depth cut of the smallest land indexed by 4 turns—10 holes, therefore a 4 turn regular index would require an increase of .018″, a 4 turn + 10 an increase of .036″, a 4 turn + 5 an increase of .027″. See diagram (8).


Reamer cutter.
Reamer.
Dead centre.
1.
2
-10
Regular 4 turns.
+5.
-5. Holes in 39 circle
+10
+10.
3.
-5.
R
+5
-10.
-10
1.
4
3
2.
4
-10.
-Cutter.
5. Land.
A.
B
Flute.
C.
A.
B.
6.
C.
10 holes.
5 "
A
7.
20 "
15 "
0 "
.018.
.009.
.036
8.
.027
.000
A

SPIRAL MILLING

Spiral milling is a very interesting operation and is used to cut spiral gears, to flute drills, etc. To do it intelligently, one should understand gear ratios thoroughly, and be able to solve the right angled triangle by trigonometry. As the cutter on the arbor of the machine rotates, the table of the machine is set at the required angle and passes the rotating cutter by means of the automatic feed. At the same time, the index head is geared to the table screw and rotates the work as it passes the cutter.

Index head set up for spiral milling. Diagrams (1) and (2). The pin at the back of the index plate is taken out of the plate, allowing the plate to rotate and the crank pin is pushed into one of the holes of the plate, so that the crank turns with the plate. This in turn rotates the worm and worm gear as in plain indexing.

Rate of rotation of index head spindle. This rate is governed by the ratio of the gears which connect the index plate with the table screw. If the ratio of the gears between the table screw and the index plate were 1 to 1, the index plate would make 1 revolution for each revolution of the table screw. Since the worm and worm gear have a ratio of 40 to 1, the spindle would only rotate 1/40 of a turn for each revolution of the table screw. The table screw has 4 threads per inch and is a single thread, so that it must rotate 40 times to cause the index spindle to revolve once. If the screw makes 40 revolutions the table will move past the cutter a distance of $40/4 = 10''$, because each revolution of the screw moves the table $\frac{1}{4}''$. This movement of $10''$ for an equal geared set up forms the basis of all calculations for spirals of different lead.

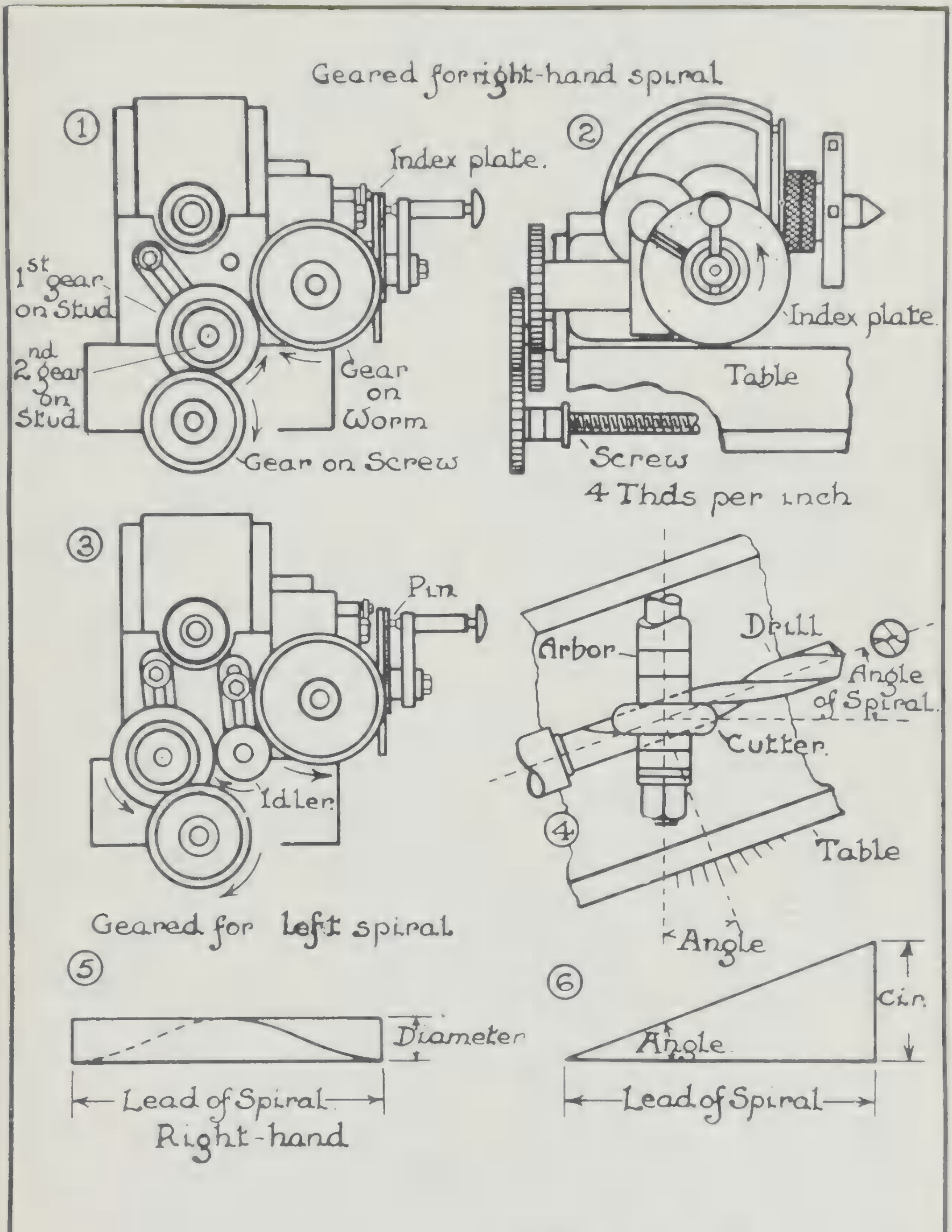
Machine equally geared for a 10″ lead spiral. Diagram (1) and (2).

If gear on screw has 30 teeth
First gear on stud has 40 teeth
Second gear on stud has 20 teeth
Gear on worm has 60 teeth

The worm will rotate once for each revolution of the screw, because $30 \times 40 = 20 \times 60$.

To find gears for a given lead. It is required to cut a spiral with a 36″ lead.

Required lead $= 36''$
Lead of machine $= 10''$
Gear ratio equals 10 to 36
Factorize $10 = 2 \times 5$ (drivers)
and $36 = 3 \times 12$ (followers)



SPIRAL MILLING.

$2 \times 10 = 20$ $3 \times 10 = 30$	First pair of gears
$5 \times 5 = 25$ $12 \times 5 = 60$	Second pair of gears
Gear on screw = 20 1st Gear on stud = 25	Drivers
2nd Gear on stud = 30 Gear on worm = 60	Followers

Proof $10'' \times 30 \times 60 =$ Required lead $\times 20 \times 25$

$$\text{Lead} = \frac{\overset{2}{\cancel{10}} \times \overset{6}{\cancel{30}} \times \overset{3}{\cancel{60}}}{\cancel{20} \times \underset{5}{\cancel{25}}} = 36''$$

To cut right or left hand spiral. The set up in Diagrams (1) and (2) will produce a right hand spiral. Diagram (3) shows the same gear ratio with the inclusion of an idler gear. The effect of this idler will only be to change the direction of rotation of the worm, making it suitable for cutting a left hand spiral.

Setting the table to the angle of the spiral. Diagram (4) shows the table swivelled to the angle of the spiral, which is right hand. For a right hand spiral the table is swung to the right and for a left hand spiral, it is swung to the left.

To find the angle of the spiral. Diagrams (5) and (6). Diagram (5) shows a cylinder with the spiral line traced around it. If this were a piece of paper wrapped around the cylinder, when it was stretched out it would be as in diagram (6), a right angled triangle with the side opposite the angle equal to the circumference of the cylinder and the side adjacent equal to the length of the cylinder and the lead of the spiral.

To find the angle of spiral, given diameter of work and lead.

Example. Diameter of work 1″, lead of spiral 8.63″. *Find* angle of spiral (circumference=3.1416″).

$$\text{Tan of angle} = \frac{\text{side opposite}}{\text{side adjacent}}$$

$$\text{Tan of angle} = \frac{3.1416''}{8.63''} = .3640.$$

$$\text{Tan } .3640 = 20^\circ$$

DIFFERENTIAL INDEXING (METHOD)

Differential indexing makes it possible to obtain a greater number of divisions than can be obtained by "Rapid" or "Plain Indexing". Owing to the fact that the gear on the worm is geared to the index head spindle gear, it is impossible to use this method of indexing for spiral work, because the worm would then be connected by gears to the table lead screw.

Details of the index head set for differential indexing. Diagram (1). The freehand sketch in diagram (1) shows in skeleton form the gears, etc., used in differential indexing. This picture form of diagram is intended to simplify the mechanism and allow one to trace through the varying rotation from the index crank to index head spindle. The latch pin at the back of the index plate (E) is removed to allow the plate to rotate freely as in spiral cutting and indexing. *Note.* ((A) shows the holes in the spindle plate used for rapid indexing). If the crank pin (D) is pulled out from a hole in the plate (E) and the crank is rotated clockwise, a clockwise rotation is given direct to the worm (B), which has a right hand thread. As the worm (B) rotates, it turns the worm gear (C) away from the operator, and turns the index spindle in the same direction.

At the end of the index spindle, a gear spindle (F) fits inside to a tapered seat, and carries a gear which connects through other gears to the gear on the worm (G). All these gears may be changed to vary the ratio and do not belong to the fixed mechanism of the index head.

The spiral gear (1) receives the rotation from the gear on the worm shaft (G) and transmits the rotation at right angles to a second spiral gear (2), which is connected through a spindle to a spur gear (3) which meshes with a spur gear (4), which meshes with a spur gear (5), which is fastened to the sleeve (H), to which the index plate (E) is fixed. The index plate (E), sleeve (H) and gear (5) all rotate on the spindle which is turned by the crank, and which operates the worm directly.

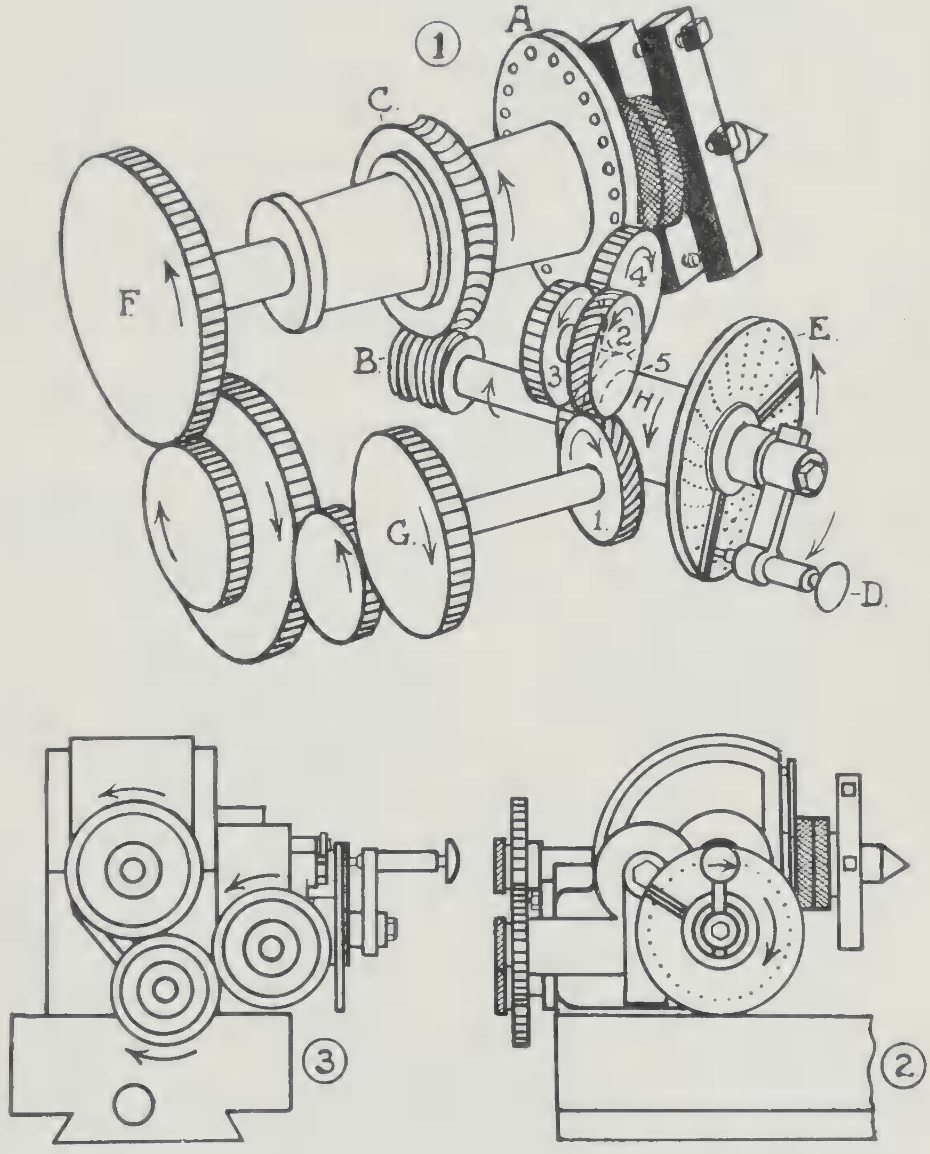
From the foregoing description, it can be seen that when the crank is turned, the index plate may rotate at any rate in the same direction or it may rotate at any rate in the opposite direction, according to the number of gears used, and their ratio.

By tracing through the direction of rotation, in diagram (1), it will be seen that the index plate will rotate in an opposite direction to the rotation of the crank. This means that instead of the index pin meeting the proper hole for the index as in plain indexing, with the plate stationary, it will meet that position earlier, so that it will not be turned as much as in plain indexing, which means that it will index

for a greater number of divisions than that for which it is set, as figured for plain indexing.

Diagrams (2) and (3) show the front and end elevations of the index head set up for differential indexing with an odd number of gears. If this rotation is traced through, it will be noted that the index plate rotates in the same direction as the crank when turned clockwise. This means that the crank pin must be turned further than would be necessary as in plain indexing with the plate stationary, because the plate now rotates with it and the crank must be turned more than usual for a number of divisions as for plain indexing to catch up to its proper hole. The effect of this will be to index for a smaller number of divisions than would be cut if the plate were stationary.

Diagram (1) shows a compound train of gears being used with one idler; and Diagrams (2) and (3) show a simple train of gears being used with one idler.



DIFFERENTIAL INDEXING.

DIFFERENTIAL INDEXING (CALCULATIONS)

If an index head is geared up for differential indexing with the gears giving a ratio of 1 to 1, the index plate will make one revolution for each revolution of the index head spindle. It requires 40 turns of the crank to turn the spindle once. If the crank pin is located in a certain hole of the plate and the relation of the pin and the hole is observed, it will be found that when the plate turns in the same direction as the crank, the crank rotates 41 times to meet the hole, and turn the spindle once. If the plate turns in the opposite direction to the crank, it will be observed that the crank rotates 39 times to meet the hole, and turn the spindle once. This relation is the principle applied in differential indexing.

Example. It is required to index 233 divisions, which cannot be obtained by plain indexing.

Solution. (1) Select a number near the required number, which could be obtained by plain indexing. *Selected number* 240.

(2) Find the difference between the selected number and the required divisions. (240—233=7). This number represents the number of divisions which must be lost in one revolution of the spindle (because the plate rotates with the crank).

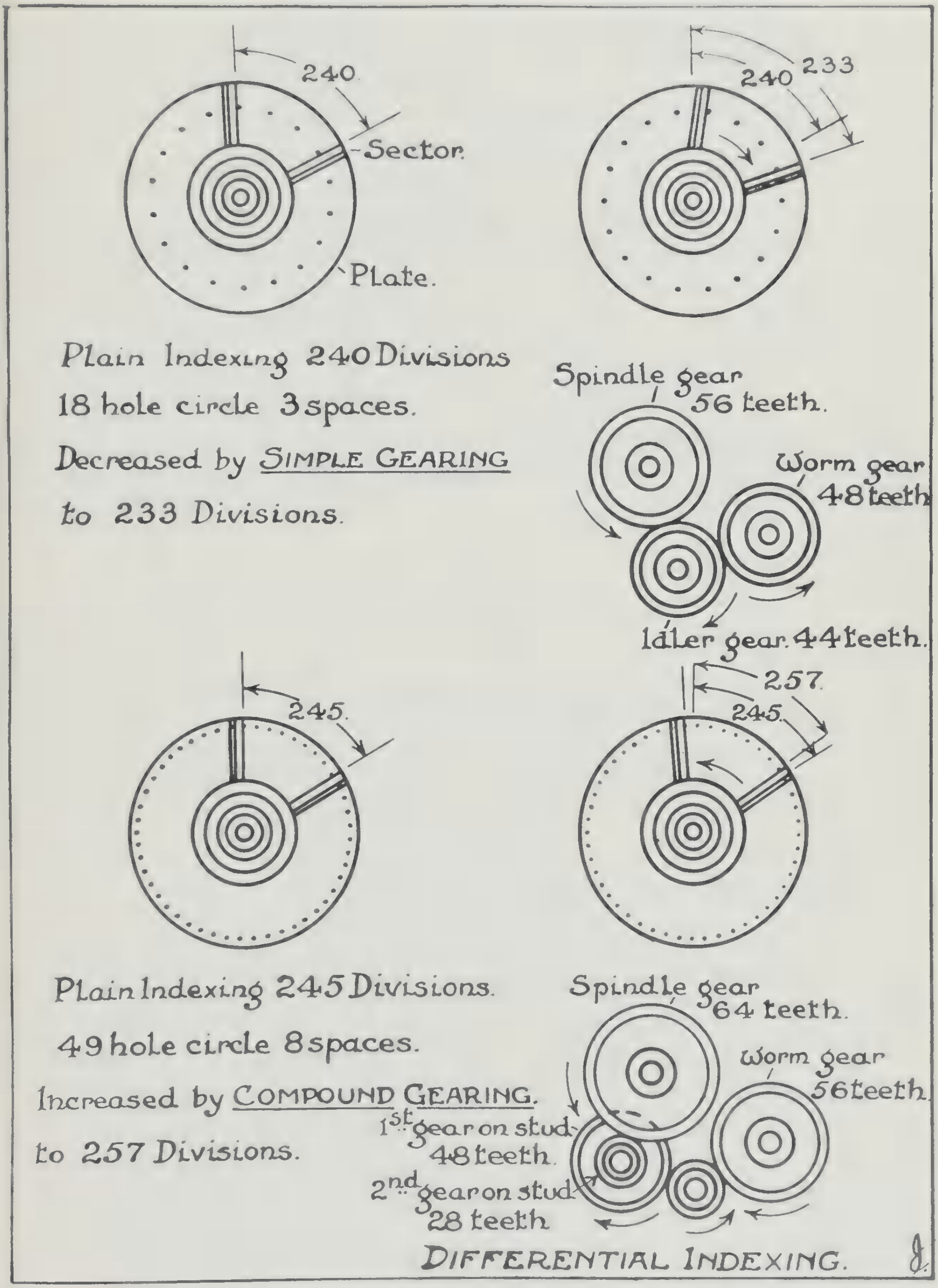
(3) Arrange a fraction with the numerator equal to the number used as in plain indexing, and the denominator equal to the number of divisions which are to be lost. Example=$\frac{240}{7}$.

Divide this fraction by 40

$$\text{Example.}\quad \frac{240}{7} \div \frac{40}{1} = \frac{240}{7} \times \frac{1}{40} = \frac{\overset{6}{\cancel{240}}}{\underset{7}{\cancel{280}}} = \frac{6}{7}\ \begin{matrix}\text{Follower}\\ \text{driver}\end{matrix}$$

This fraction 6/7 equals the gear ratio, the numerator (6) representing the gear on the worm and the denominator (7) representing the gear on the spindle. (Select gears 48 and 56, ratio 6 to 7).

(4) The gear on the worm and the gear on the spindle must turn in the same direction when less divisions are required and in opposite directions if more divisions are required. In the example given, less divisions are required (233 is less than 240); therefore the gear on the worm and the gear on the spindle must rotate in the same direction. To accomplish this an idler must be used. Diagram (3).


240.
240
233
Sector.
Plate.
Plain Indexing 240 Divisions
18 hole circle 3 spaces.
Decreased by SIMPLE GEARING
to 233 Divisions.
Spindle gear
56 teeth.
Worm gear
48 teeth
Idler gear. 44 teeth.
245.
257.
245.
Plain Indexing 245 Divisions.
49 hole circle 8 spaces.
Increased by COMPOUND GEARING.
to 257 Divisions.
Spindle gear
64 teeth.
Worm gear
56 teeth.
1st gear on stud
48 teeth.
2nd gear on stud
28 teeth
DIFFERENTIAL INDEXING.

To find index for 233 divisions. Diagrams (2) and (1).

(1) Index for 240 divisions as in plain indexing. Diagram (1).

$$\text{Ratio } \frac{40}{240} = \frac{1}{6}$$

(2) Select an 18 hole circle.

$$\frac{1 \times 3}{6 \times 3} = \frac{3}{18} \quad \begin{matrix}\text{(spaces)} \\ \text{(hole circle)}\end{matrix}$$

(3) Index for 240=3 spaces in an 18 hole circle.

(4) Decrease this by simple gearing to 233 divisions. Diagram (2).

$$\text{Gear ratio used} = \frac{6 \times 8}{7 \times 8} = \frac{48}{56} \quad \begin{matrix}\text{gear on worm} \\ \text{gear on spindle}\end{matrix}$$

Connect up with an idler as shown in Diagram (3).

Differential indexing using a compound gear train. *Question.* Index for 257 divisions (diagram 5). *Solution.* Select 245 divisions as for plain indexing (diagram 4).

$$\frac{\overset{8}{\cancel{40}}}{\underset{49}{\cancel{245}}} = \frac{8}{49} \quad \text{Index 8 spaces in a 49 hole circle}$$

Difference between selected number and required divisions: 257—245=12 (divisions to be gained).

$$\frac{245}{12} \div \frac{40}{1} = \frac{245}{12} \times \frac{1}{40} = \frac{\overset{49}{\cancel{245}}}{\underset{96}{\cancel{480}}} = \frac{49}{96}$$

Factorize. $\dfrac{49}{96} = \dfrac{7 \times 7}{12 \times 8} \text{ gears} = \dfrac{28 \times 56}{48 \times 64} \quad \begin{matrix}\text{followers} \\ \text{drivers}\end{matrix}$

Set up gears as shown in diagram (6) with idler to give opposite rotation to the index plate compared to the clockwise rotation of the index crank (diagram 5), so that the required hole comes to meet the index pin to give a lesser index, and a correspondingly greater number of divisions.

QUESTIONS ON MILLING

1. Explain the difference between "Direct" and "Plain Indexing".
2. What divisions can be indexed by "Direct Indexing?"
3. Can *all* divisions be obtained by "Plain Indexing?" What other method of indexing can be used?
4. Make a freehand sketch of the mechanism used for "Plain Indexing." What is the ratio of this mechanism?
5. Should the Index plate be locked or free to rotate when:—Plain Indexing? When must it be free to rotate?
6. What must be done to the Plain Indexing mechanism before rapid indexing can be carried out?
7. Why is it necessary to use the "Sector Arms" when plain indexing?
8. How many turns of the crank will be used to index 17 divisions on work? What will the sector arms be set at?
9. How would you index for 23° on work?
10. Sketch an end view of a reamer being cut on a milling machine, showing the position of the cutter with regard to the work.
11. Why is it necessary to cut a reamer by indexing with an irregular number of holes?
12. State two methods by which the lands of an irregularly spaced reamer could be made equal in size.
13. Tabulate the indexing of a reamer having 10 flutes with cutting edges diametrically opposite and irregularly indexed.
14. Sketch the end view of a reamer showing the cutting of the first four flutes; number each flute in the order in which it is cut.
15. How would you find the depth increment for correcting the land sizes of an irregularly cut reamer?
16. Describe the set up of a milling machine to cut a spiral gear.
17. What is the basis of calculations for the gear ratios when spiral milling?
18. Sketch a right hand spiral.
19. Is the table turned clockwise or counterclockwise to cut a right hand spiral?
20. Sketch the stretchout of a spiral showing the circumference, lead of spiral, and the angle to which the table is swivelled.
21. Describe the movements of the mechanism of an Index Head set up for differential indexing.
22. What do the change gears of a differential "set up" on an index head effect?
23. Why is differential indexing necessary?
24. If the change gears on a differential index "set up" have a 1 to 1 ratio, how many turns will the index plate and spindle each make to each revolution of the crank?

25. If the crank pin of an index head is located in a certain hole, how many times must the crank be turned to turn the spindle round once (a) when the plate rotates in the same direction as the crank? (b) when the plate rotates in the opposite direction to the crank?
26. How is the gear ratio for differential indexing found?
27. In what direction must the gear on the worm and the gear on the spindle rotate:

    (a) When less divisions than the selected number are required?

    (b) When more divisions than the selected number are required?

    Note:—For questions on the mathematics of milling, see mathematics section.

GRINDING

ABRASIVES USED IN GRINDING AND POLISHING

Abrasives have been used for centuries for sharpening tools and shaping metal, but most of the development in manufacture and use of abrasives has occurred during the last forty years. Such companies as the "Norton Grinding Company" and the "Carborundum Company" have done much to develop modern grinding in industry. These companies have given freely of their knowledge and by writing to them, students can obtain a full line of literature which will be of great assistance to them in understanding the grinding field.

Classes of Abrasives. Abrasives may be classified into two main divisions. (a) Natural abrasives; (b) Artificial abrasives.

Sandstone, emery and corundum are natural abrasives; that is, they are found ready made in the earth. The old familiar grindstone, made by forming the wheel from sandstone rock and mounting on a spindle, will be remembered by many. It is much softer than our modern abrasives and is not regular in hardness, which causes it to wear unevenly and therefore wears out of shape.

Emery contains approximately 50% to 60% crystalline aluminum oxide, and most of the remainder is iron oxide.

Corundum contains approximately 75% to 90% crystalline aluminum oxide, and is harder than emery.

Natural abrasives vary considerably in quality so that artificial abrasives are mostly used to-day because they can be produced with even quality and temper; but natural abrasives are still preferred for some few operations.

Artificial abrasives may be divided into two main groups.

(1) Aluminum oxide abrasives.
(2) Silicon Carbide abrasives.

Aluminum oxide abrasives are sold under various trade names. The Norton Company calls their abrasive of this type "Alundum" and the Carborundum Company calls theirs "Aloxite" and other companies have other trade names under which this abrasive is sold.

This abrasive is made chiefly from a raw material called "Bauxite" a mineral found in Arkansas, U. S. A. The principal content of Bauxite is Aluminum oxide combined with water and oxides of iron.

The Bauxite is melted in the electric furnace and produces a very hard substance similar in properties to Emery and Corundum but more uniform in grade.

Aluminum oxide abrasives vary in color, they are usually brown, but other types are made white or gray in color. The crystals vary in size

in the mass, some being as large as one quarter of an inch to one inch in diameter. They are not as hard as Silicon Carbide abrasive but are much tougher. This abrasive is used for metals that are tough or hard and have great tensile strength.

The method of manufacture can be used to control the temper of the abrasive required, to suit various classes of work to be ground. The abrasive grains are prepared by breaking up the mass of abrasive crystals so that the grains are fragments of the original crystals and are screened to obtain various sizes of the grains.

Silicon carbide is sold under various trade names. It is called "Crystolon" by the Norton Company, and "Carborundum "by the Carborundum Company.

Silicon carbide is composed chiefly of the elements Silica (sand), and carbon, and is made from the following ingredients: coke, sand, sawdust and salt. It is made in the electric furnace and is very crystalline, having a greenish blue color and is iridescent in mass form. The crystals are much harder than aluminum oxide abrasives, but very brittle. When the crystals break, they present an angular form with sharp cutting edges. It is found a very suitable abrasive for grinding metals or materials of low tensile strength such as cast iron, brass, bronze, aluminum, marble, etc.

The properties of an abrasive. An abrasive is a tool with a cutting edge the same as other cutting tools and is able to cut away materials which are softer than itself. The abrasive has a hardness which allows it to cut metals, and a toughness which allows it to hold together under the strain of cutting. The abrasive when broken under strain breaks with a fracture that provides other cutting edges. Artificial abrasives can be made with varying temper, which means that the strength of the abrasive may be varied, to break up during a light load of cutting or a heavy load, this deals with the abrasive itself and not the bond which holds the abrasive grains together in a grinding wheel.

Manufacturing grinding wheels. If some abrasive grains are examined with the object of imagining how these grains may be held together to hold to some definite shape, one can visualise the similarity of the problem of a child building sand castles. The child finds that wet sand holds together better than dry sand, in other words the water becomes the *bond* to hold the particles together.

Abrasive grains have to be held together to form grinding wheels and stones of various shapes. When one realises that these wheels run at a peripheral speed of 5,000 to 6,000 ft. per minute it is easy to see that the bond must be strong, to prevent the wheels from flying apart. The making of wheels with abrasive grain is essentially a potter's job, using as he does various clays, etc., to form the bonding material to shape the mass of the wheel form.

The processes of wheel manufacture are: Vitrified, Silicate, Elastic and Vulcanized or rubber.

The vitrified process uses a bond of flux and clay mixed with the abrasive and fused at high temperature in an electric furnace. The clays and fluxes and water are mixed with the abrasive to a muddy consistency and run into molds.

After setting to the form of the mold it is turned into a wheel form to the correct size and shape then baked and afterwards placed in the electric furnace at about 3,000°F for about 100 hrs. which causes the clay to be vitrified into a hard glassy-like bond which has porosity, but holds the abrasive grain firmly together. Approximately 80% of all grinding wheels are made by the vitrified process.

Porosity is an important thing in grinding wheels as it allows room for the chips of metal removed when grinding, and makes the wheels free cutting.

Grade of the wheel is dependent upon the bond. There are many combinations of clays used to determine the grade of the wheel, and finer grade distinctions depend upon the quantity of bond. Most grading is done by hand, and requires considerable experience on the part of a grader.

The grading tool is like a short screwdriver with a sharp thin edge which is pushed against the wheel face and turned slightly to test the resistance offered by the wheel.

Grade Table (Norton Company)

	Bond	Very soft	Soft	Medium	Hard	Very hard
(1)	Vitrified	E.F.G.	H.I.J.K.	L.M.N.O.	P.Q.R.S.T.	U.W.Z.
(2)	Silicate					
(3)	Elastic	½	1, 1½, 2	2½, 3, 4	5, 6,7	
(4)	Rubber				R. S. T.	U. W. Z.

Grade Table (Carborundum Company)

	Bond	Very soft	Soft	Medium	Hard	Very hard
(1)	Vitrified	W.V.U.	T.S.R.P.O.N.	M.L.K.J.I.	H.G.F.	E.D.
(2)	Silicate	W.V.U.	T.S.R.P.O.N.	M.L.K.J.I.	H.G.F.	E.D.
(3)	Shellac	10	9.8.7.6.	5.4.3.	2.1.	
(4)	Redmanol	17	16,15,14,13,12,11	10,9,8,7,6 F.E.	5,4,3 D.C.B.	

Standard Grain Sizes (Carborundum Company) for Carborundum and aloxite

Very Coarse	Coarse	Medium	Fine	Very fine	Powders commercial grading	Powders Microscopic grading
6	12	30	70	150	F.	280
8	14	36	80	180	F.F.	320
10	16	40	90	220	F.F.F.	400
	20	50	100	240		500
	24	60	120			600

Processes of wheel manufacture

Vitrified	Silicate	Elastic	Rubber
1. Porous-free cutting. 2. Bond-hard, strong. 3. Uniform—no soft spots. 4. Not affected by acids, water, oils, heat, cold. 5. Made in Electric furnace at (3,000 F.) for 100 hrs. 6. No Elasticity. 7. Used for ¾ of all grinding.	1. Not free cutting. 2. Bond-Silicate of soda. 3. Hard and soft spots may occur. 4. Molded and baked low heat. (500 F.) 5. Safe. 6. Wheels made any size or on iron backs or wire webbing.	1. Not so porous. 2. Bond-Shellac. 3. Thin wheels may be used. 4. Pressed into molds—baked few hours 300 F. 5. Process rapid. 6. Wheels can be made on iron backs. 7. Good for brass.	1. Hard, tough 2. Bond-Rubber. 3. Very thin wheels. 4. Sheets of rubber and abrasive grain in layers then "died out" and vulcanized.

PROFILE OF LATHE TOOLS

This lesson is one of a series dealing with some of the particular features underlying the science and art of tool grinding. If an operator understands the effect of various shapes and angles on the work being machined, he will grind tools more intelligently. The tendency to-day is to have all tool grinding done by an operator on a special tool grinding machine using standard shapes and pre-determined angles, but there are many shops where it may be necessary for workmen to grind their own tools, and the basic principles presented here and in other lessons to follow are offered as fundamental guides.

The shape or profile of the tool is illustrated in the adjacent diagrams, and various considerations will be presented dealing with the shape of the top face of the tool, showing the effect it has on its efficiency as a cutting instrument. Diagrams (1A) and (1B) show two different tools doing the same work, with equal depth of cut and equal feed. The tool (1A) takes less time to grind than tool (1B) because less metal is removed. The tool point in (1A) is supported by more metal than tool (1B): therefore it is obvious that under the chip pressure when the friction generates heat it will be carried away from the cutting edge by the mass of metal behind it, thus dissipating or using it up and keeping the tool comparatively cool.

Tool (1B) has a weak point with very little support, and owing to the metal being ground off as shown by the dotted line it will not dissipate the heat as well as tool (1A) so that the point heats up quickly and soon requires regrinding.

Diagrams (2A) and (2B) show two tools turning cast iron which has a hard skin. The shaded portions in each case show the area of the tool under the pressure of the hard skin. Tool (2A) has less wear on the cutting edge because of the lessened area in contact; therefore it will last longer than tool (2B). The point being the weakest part of the tool should always cut beyond the skin into the soft metal beneath, as shown in the sketches.

Diagrams (3A) and (3B) show two tools cutting at equal depth, with equal feed. The round-nosed tool will produce smoother work than the pointed tool because the chip removed is gradually tapered off. The arrows in each case show the direction of the chip pressure. Tool (3A) is under greater pressure than tool (3B), because in addition to the ordinary chip pressure the chip is distorted so that extra work is required to do this. Tool (3B) produces a chip which is a smooth band of metal that comes from the work freely so that the advantage is with the tool with a straight cutting edge.

Diagrams (4A) and (4B) illustrate the effect of chip pressure on the work due to the angle of the cutting face. A triangle of forces is illustrated in each case; the hypotenuse of the triangle is the resultant force of the chip pressure. If it is broken up into its horizontal and vertical components it will be noted that tool (4A) has less reactive force (tending to push the work away) than tool (4B). Therefore it can be assumed that for slender work, likely to spring, a tool similar to tool (4A) should be used. Tool (4B) will wear longer than tool (4A) because the chip pressure is distributed over a greater cutting edge, but owing to the angle of the cutting face it is only suitable for heavy work.

Conclusions:

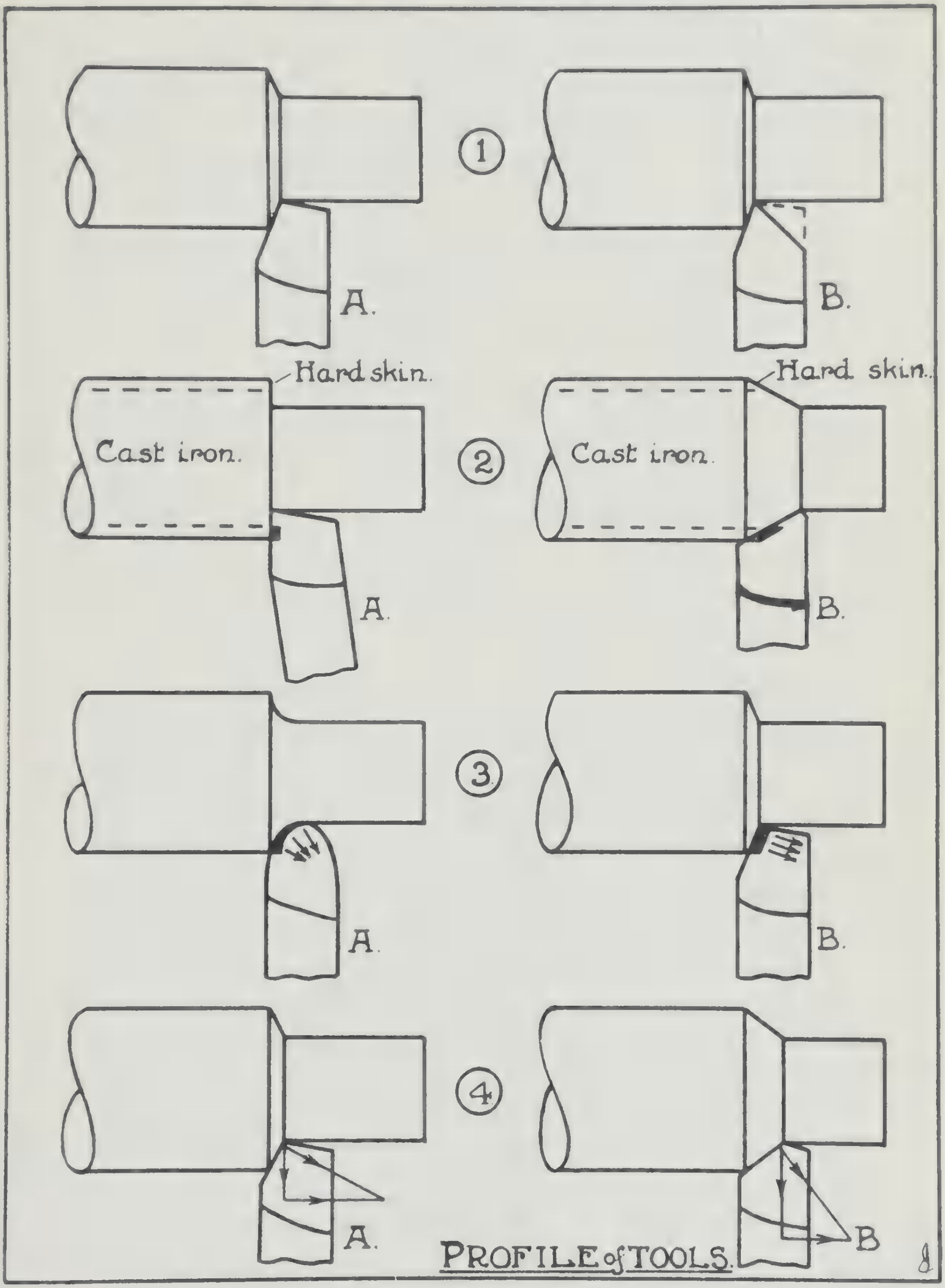
(1) When grinding a tool do not remove more metal than is absolutely necessary.

(2) A tool with a rounded point produces smoother work than one with a sharp point.

(3) Cut the skin of cast iron with the cutting face of the tool almost at right angles to it, and keep the point of the tool beneath the skin in the soft metal.

(4) A straight-edged tool cuts more easily than a round-edged tool.

(5) The angle of the cutting face depends upon the diameter of the work.



PROFILE of TOOLS.

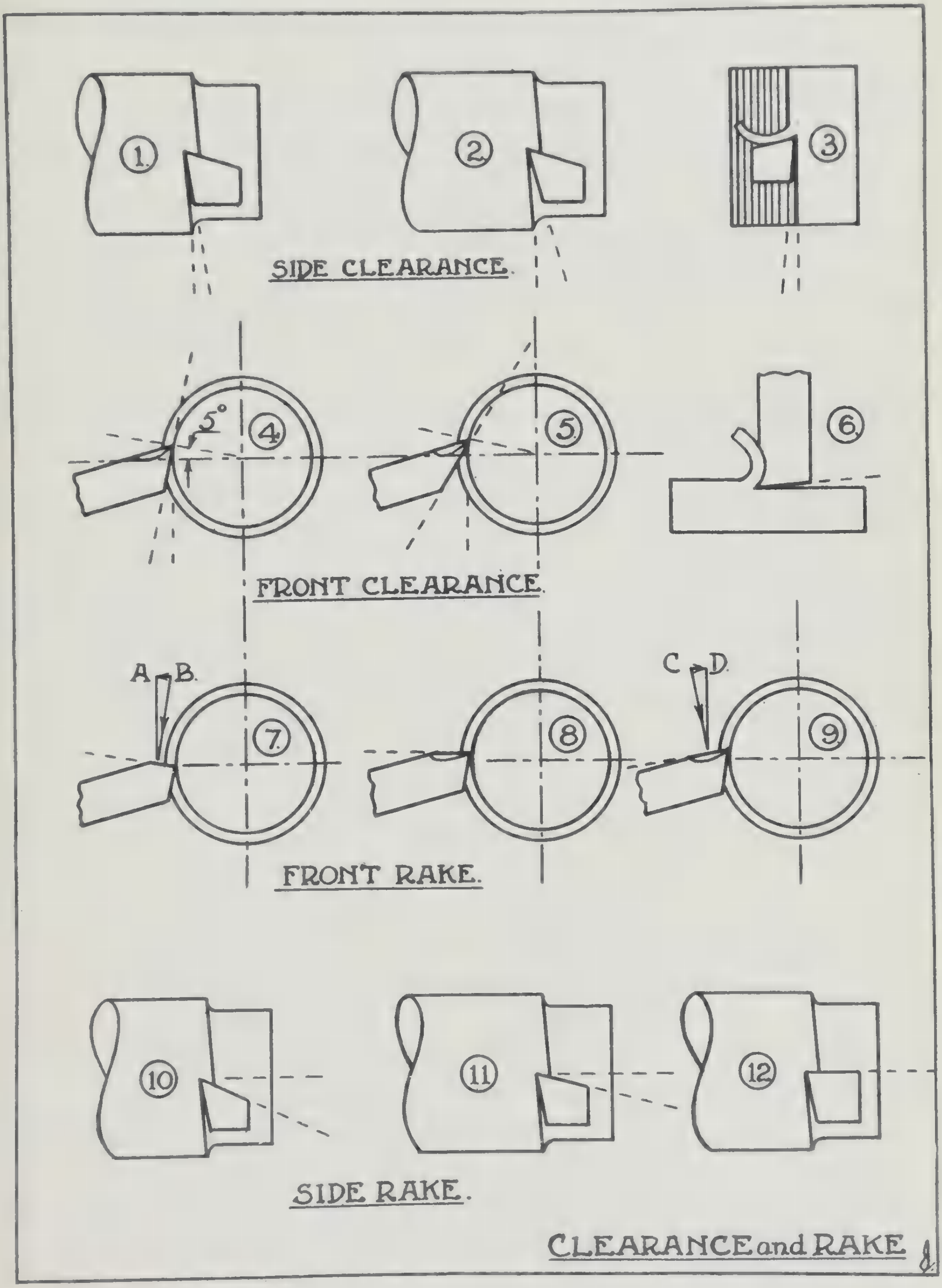
CLEARANCE AND RAKE

This lesson deals particularly with the effect of grinding angles on the tool bit which are commonly known as side clearance, front clearance, front rake and side rake. It is not necessary for every tool to have all such angles ground on them, but the fundamental principles introduced here will help to give a grinding operator some direction as to the purpose of such angles, when to use them, and their influence on the efficiency of the tool.

Side clearance: Diagram (1) shows a tool with a side clearance of 6°, which has been proved to be ample for turning work at ordinary feeds. As long as the side face of the tool does not rub against the work it will be most effective when the clearance angle is as small as possible. By comparing diagram (1) with diagram (3) it will be seen that a shaper tool has a similar clearance, the difference being that the lathe tool has a clearance on the angle of the work being cut, due to the rate of feed, while the shaper tool clearance is only such as to clear a straight edge parallel to the direction of the progress of the tool. Diagram (2) shows a tool with too much clearance, which causes weakness in the tool with a tendency to dig into the work beyond the rate of the feed, probably causing a chattered surface on the work.

Front clearance: The shaper tool in Diagram (6) requires about 6° or 7° to prevent it from rubbing the work and overheating the tool so that the lathe tool which cuts a receding surface of the work does not require a clearance that is too great. The position of the height of the tool with regard to the centre of the work is important. Diagrams (4) and (5) show two tools placed at 5° above the centre of the work, which is about the best position for the usual clearance angles to give the best results. The front edge of the tool should be tangent to the finished diameter of the work being machined, as shown in diagram (4). Diagram (5) shows a tool with a clearance that is too great. This tool does not give sufficient support to the cutting edge and tends to dig into the work, as drawn by the dotted line.

Front rake has many influences on the cutting action of the tool. First, it provides a natural passage for the chip. It therefore influences the shape and pressure of the chips as they leave the work. Second, too much rake weakens the cutting edge, which may cause the tool to lose its keeness. A tool with rake requires less power than one without rake. Third, as the chip rubs on the tool face, it exerts a pressure on it and the influence of the pressure either tends to pull the tool into the work or to push it away from the work. Brass has a natural tendency to draw tools


1
2
3
SIDE CLEARANCE.
5°
4
5
6
FRONT CLEARANCE.
A
B.
C
D.
7
8
9
FRONT RAKE.
10
11
12
SIDE RAKE.
CLEARANCE and RAKE

into itself, therefore the top face should have a negative rake, as shown in diagram (7). The line A. B. represents the horizontal component of the force due to chip pressure on the face, which tends to push the tool away and keep it from drawing into the work.

Diagrams (7), (8) and (9) show 3 tools, each suitable for cutting different kinds of metal. Soft and tenacious metals, like very soft steel require a keen rake. Metals tough and difficult to cut, like tool steels and most alloy steels, require very little rake. The rake varies with the extreme nature of metals. Very few metals, including most brasses and chilled cast iron, require no rake at all.

Diagram (8) shows a tool with no front rake. It is suitable for cast iron and some of the steels. Diagram (9) shows the influence of front rake. When metal resists penetration but is not too brittle, the chip pressure tends to assist the tool to penetrate the work, as shown by the horizontal component C. D. of the chip pressure on the top face of the tool.

Side rake provides a passage for the chip as the tool progresses, due to the effect of the feed. What has been said about front rake applies equally well to side rake. Diagrams (10), (11) and (12) show varying degrees of side rake which may be given to tools, according to the kind of metal being cut. The side rake relieves the pressure on the tool at the expense of the strength of the tool. Tool (10) would be used for soft metal, tool (11) for medium hard metal and tool (12) for very hard metal.

CUTTING ANGLES

The two common types of lathe tools are the solid or forged type and the tool bit and toolholder type. The solid tool, which is shaped from one piece of steel, is rigid and dissipates heat well because of its bulk as compared to the tool bit and toolholder. The tool bit and toolholder, however, tend towards economy of high speed steel and offer possibilities of using many-shaped cutters in the toolholder.

The toolholder, as illustrated, provides an inclination for the tool bit and also a front rake equal to the angle of the slot in the toolholder, which is usually 15°. The inclined slot also offers a method of raising the height of the cutting edge in addition to the usual method provided by the rocker in the base of the toolpost.

The tool bit is made of the various brands of high speed steel, or possibly "Stellite" or "Tungsten Carbide", and is sold in suitable lengths with the ends cut at an angle to save time in grinding for clearance. Cemented Tungsten Carbide is very expensive and a small piece of this metal is brazed to a steel tool bit to form the cutting edge. It is important to remember, when grinding a tool-bit, that it will be held at an angle when in use in the toolholder, therefore this angle must be allowed for when reckoning the front clearance and front rake. The side clearance and side rake are the same as for a solid tool because the inclination in the toolholder does not affect these angles. The upper left diagram shows a tool bit as it would rest on a block for testing its angles. It will be seen that the 30° front clearance, when placed in its used position as shown in top right diagram, becomes a 15° front clearance. Similarly no front rake in the left horizontal diagram becomes a 15° front rake in its used angular position.

Side clearance: The six central diagrams show that the side clearance for the cutters for various metals is the same. The reason for this is that side clearance is chiefly governed by the rate of feed; and 6° has been proved to be ample for ordinary feeds. If the tool does not rub the work the side clearance is satisfactory. Too much side clearance weakens the tool, does not offer sufficient support to the cutting edge and lessens the heat-dissipating ability of the tool.

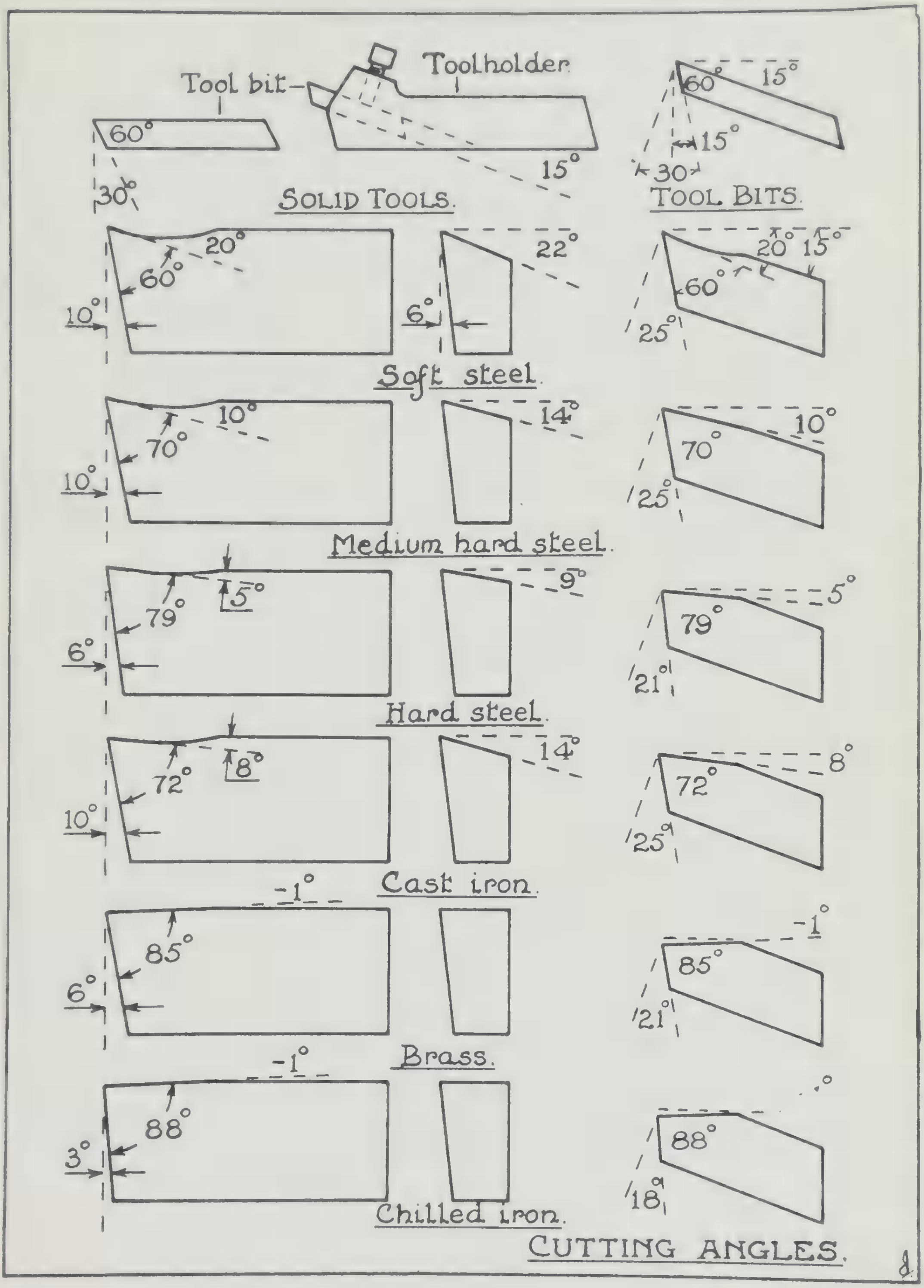
Solid tools: The six left diagrams show typical tools for various metals. As a general rule, the cutting angle is less for soft metals and more for hard metals (brass is an exception). The most important angles in any tool are front rake and side rake as they affect the chip pressure and the ability of the tool to do its work with the least strain and still preserve its cutting edge.

The solid tool is rectangular in section while the tool bit is square in section, so that it is obvious that for heavy cutting the solid tool is superior to the tool bit. The solid tool which is forged to shape has its drawbacks, because when it is worn it is necessary for it to be re-forged and hardened and tempered.

Tool bits are provided already hardened and only need grinding to the shape required. It is not wise to grind below the top edge of the tool more than is necessary, because if the top is continually ground it will be necessary to grind off the whole of the ground lip of the tool and re-grind a new point. When re-sharpening a tool bit, be careful to grind the front to provide a keen edge so that the top face is not ground down too low and thus prevent grinding off the whole point through careless handling.

Clearances: The clearances illustrated in the diagrams are the minimum for standard shop tools ground by a trained grinder, or ground on an automatic grinding machine. If workmen grind their own tools, unless they have experience in that operation, it is far safer for them to increase the clearances rather than decrease them. Some workmen grind angles by looking at them without gauges of any kind and for this guessing method side clearance angles for example would be better nearer 10° than 6°.

Front clearance varies, of course, with height setting of the tool. The higher the point is set with regard to the centre of the work the less the relative clearance angle will be. The front clearance should be sufficient so that no other part but the cutting edge touches the work. Its other influence is that it helps to govern the cutting angle of the tool, thus influencing the support to the cutting edge.


Tool bit
Toolholder
60°
30°
15°
60°
15°
15°
30°
SOLID TOOLS.
TOOL BITS.
20°
60°
10°
22°
6°
20°
15°
60°
25°
Soft steel.
10°
70°
10°
14°
10°
70°
25°
Medium hard steel.
5°
79°
6°
9°
5°
79°
21°
Hard steel.
8°
72°
10°
14°
8°
72°
25°
Cast iron.
-1°
85°
6°
-1°
85°
21°
Brass.
-1°
88°
3°
88°
18°
Chilled iron.
CUTTING ANGLES.

THE THEORY OF CUTTING

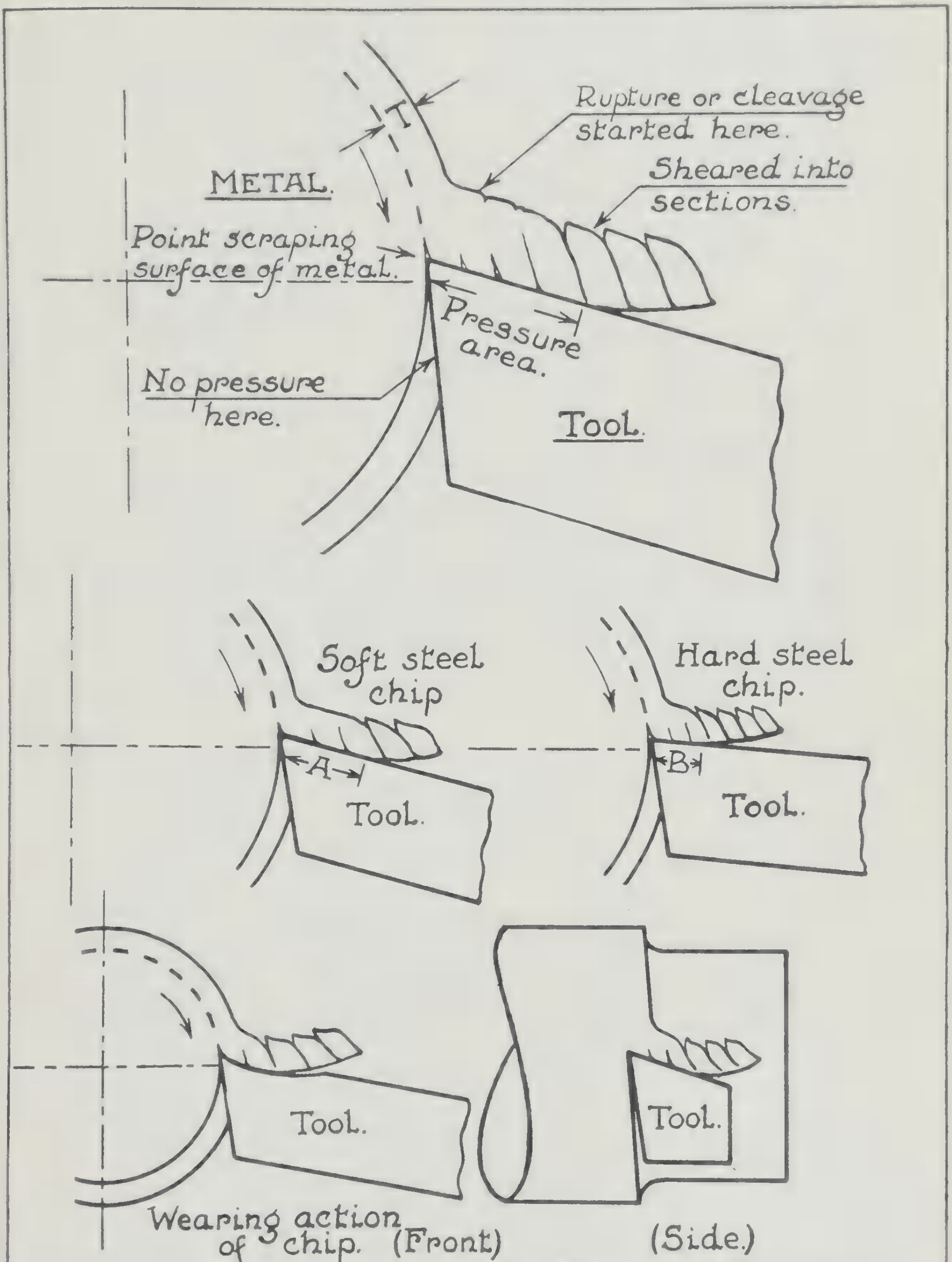
This lesson is intended to present considerations of how cutting occurs in turning work. The work rotates past a stationary tool, the point of which is set for a depth cut and the tool is forced by the feed mechanism to progress in a line parallel to the axis of the work in rotation. The questions one might ask are: "Just what does the tool do?" "How is the chip removed?" "What influence does the chip removal have on the work itself and on the tool that removes it?"

An elementary conception of cutting generally is that a wedge is being forced into material to split it apart. This is wholly true of some tools and only partly true of others. One can see the similarity between knives, scissors, lawn mowers, razors, wood planes and wood chisels, axes, metal lathe tools, planer tools and milling cutters. The shape behind each cutting edge, whether used stationary or in rotation, is primarily a wedge, but all do not act as wedges. A knife cutting a piece of wood has both sides of the knife blade in contact with the material, one side bearing against the body of the wood and the other side forcing the chip or shaving away.

If reference is now made to the enlarged drawing, it will be observed that the front edge of the tool does not and should not bear against the metal at all. The point or extreme edge of the tool does have contact with the metal, and the function of this contact is chiefly to scrape and cut the surface of the metal after rupture, and clean up all the torn particles of metal thus leaving the metal with a smooth surface. The big work of the tool in the removal of the chip occurs on the lip of the tool just back of the cutting edge, as shown by the pressure area in the diagram. There is no true wedge action at all; the metal is forced by rotation against an inclined surface of a hard steel tool and the chip is removed by a pressing, tearing or shearing action. After the separation has begun, the crowding action of the metal on to the surface of the tool shears the chip into sections, and the shape of the chip depends upon the angle and shape of the top surface of the tool.

It will be noticed that the chip thickness is greater than the thickness of the layer being removed, as shown by (T) in the top diagram. This is due to the piling up action of the metal as it resists removal and the piling up stops when the metal "slips" or shears into sections.

A lever action enters into the chip removal to some extent. As the chip starts to strike the top surface of the tool as shown by the pressure area, the metal covering the pressure area becomes a metallic lever, prying off other metal remaining on the body of the work being turned, and this action goes on continuously.



THE THEORY of CUTTING.

Referring to the two central diagrams, it will be noted that the chip pressure for soft metal bears on a greater surface of the tool than the chip pressure for a hard metal. Anyone who has sheared soft steel and hard steel knows that very little of hard steel is actually cut in shearing, the remainder suddenly gives way; while with a soft metal the cutting edge enters much further before the metal breaks away. This action is shown in the diagram; the hard steel section shears quickly and is consequently thinner than the soft steel shear sections of the chip.

If a tool is ground with a flat top for front rake, it will be observed that the metal does not slide away freely from the cutting edge, but tends to bite into the metal and wear a curved surface upon it. It is therefore obvious that the passing chip wants an easy passage and tries to make one for itself by wearing the top face into a curved form. This condition holds good for both front rake and side rake, as shown in the two lower diagrams.

In conclusion, it would appear that a chip is torn away from metal in rotation at a point slightly above the actual point of the tool, and this tearing action leaves the surface of the work rough and jagged; but as it comes past the point the irregularities are pushed and scraped off and the surface of the metal passes the tool comparatively smooth.

FEATURES OF SPECIAL TOOLS

The previous lessons have illustrated the effect of the various grinding angles of a tool. Some fundamental facts have been proved and it has been ascertained that there are many contrary elements to deal with in relation to tool grinding.

It is the purpose of this lesson to show that tools designed for a particular kind of work do not possess all the grinding angles already mentioned, but only such angles as will assist the tool in doing its work effectively in cutting certain kinds of metal.

Turning brass: Diagram (1). The cutting angles, clearance angles and rake angles of a brass tool are shown in diagram (7), lesson (2), page 112, and in lesson (3), page 117. This diagram (1) shows the profile of a brass tool, which is all important if good work is to be produced. If an ordinary tool were used to turn brass, it would have quite a large cutting edge in contact with the metal, such as is shown in diagram (2). The effect of the great amount of contact with the metal would be to produce a surface on the work which might be termed "wavy", and it is then difficult to produce a smooth surface. To avoid this the tool edge in contact with the metal is kept very small, so that only a small amount of metal is operated on and the tool in relation to the work is very stiff and does not give to the cut.

Pulling in action on a tool when stopped in its feed is illustrated in diagram (2). This action is very prominent when the tool has a very round nose, giving a large amount of cutting edge in contact with the work. The two arrows show that there is a pinching action between the work and the tool, which tends to draw the tool into the work and reduce the diameter in the form of a groove.

Diagram (3) shows the best round-nosed tool for ordinary turning. The cutting edge has a mean angle of about 20° with the tool bit. This does not cause undue spring of the work, and has a small round nose to produce a smooth finish.

The parting tool is usually a troublesome tool to use. Most parting tools used now consist of a high speed blade held in a special holder. This blade has no back clearance from the cutting edge, but has side clearance, as shown in diagram (5). If a parting tool is ground from an ordinary tool bit, as shown in diagram (4), it is advisable to give it a slight back clearance from the cutting edge to make it cut cool and free from the rubbing the sides of the cut. The parting tool should be set exactly on centre for two reasons. First, to cut the bar clear through,

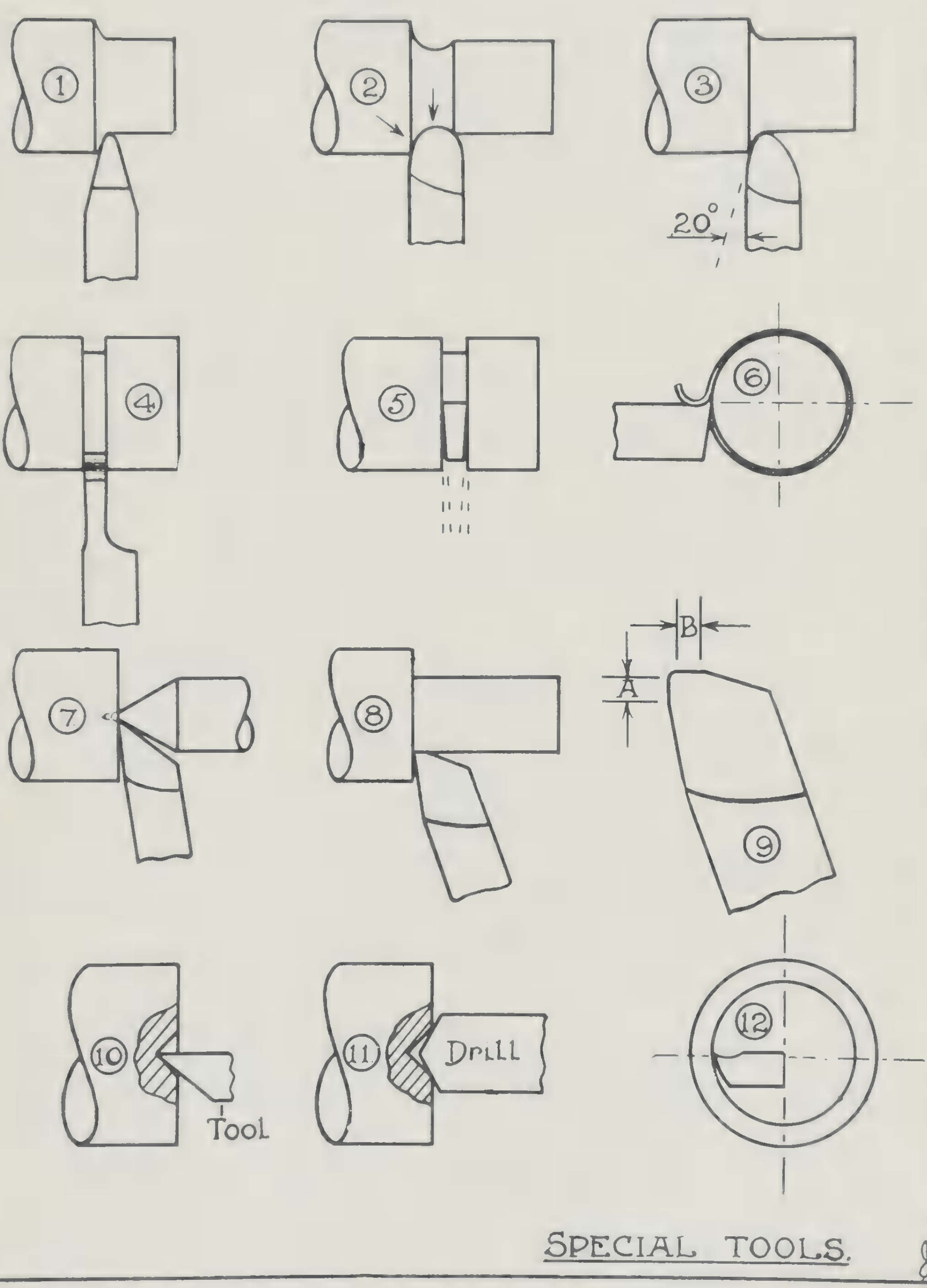
the front of the cutting edge must be on the centre of the stock, and second, if the tool is pressed down under the chip pressure, it presses away from the work, whereas if it were above centre it would be pressed down into the work—(Diagram 6).

Squaring work on centres: The tool required for this operation must be less in profile angle than 60° to enable it to do its work. Instead of having one long cutting edge it is better to have a shorter cutting edge to give less contact with the work, as a long cutting edge often causes a chattered surface on the work.

Smooth turning to shoulders: This tool is shown in the cutting position in diagram (8) and enlarged in diagram (9). The two flats A and B are at right angles to each other and the point of the tool is slightly rounded to produce smooth work both on the turned portion and on the squared shoulder.

Spotting before drilling in the lathe: When work is held in the chuck, it is necessary to obtain a small conic recess in the work, to provide a starting point for the drill, to prevent it springing off the centre of rotation of the work. The tool ground for this work must have an exceptional front clearance to allow the cutting edge only to do its work. The cutting edge is naturally weak, but this cannot be avoided consequently light cuts must be taken. A combination drill may be used in place of this tool. Diagram (11) shows the relative angles of the drill point and the conic recess made by the spotting tool. This clearance shown keeps the drill working towards the point of least resistance, which is the centre of the work.

Boring tools, Diagram (12). A tool working inside a hole must have more front clearance than a tool working on the outside of a cylinder, because the curvature of the hole is approaching the tool, whereas in ordinary turning the curvature of the work is away from the tool.



SPECIAL TOOLS.

THE TOOL AND THE CHIP

It is the purpose of this lesson to show that the form of the top face of a tool is the deciding factor in the shape of the chip produced. Most lathe operators have experienced the inconvenience caused by having long stringy chips wrap around their work. The trouble taken in removing such entanglements is an annoyance and causes a distinct loss of time and is generally inefficient.

The aim in grinding tool bits from the standpoint of the type of chip removed, should be to so grind the tool that the top face provides as easy passage for the chip and yet curls it in a convenient form, or better still curls it and breaks it up so that it will drop clear of the rotating work.

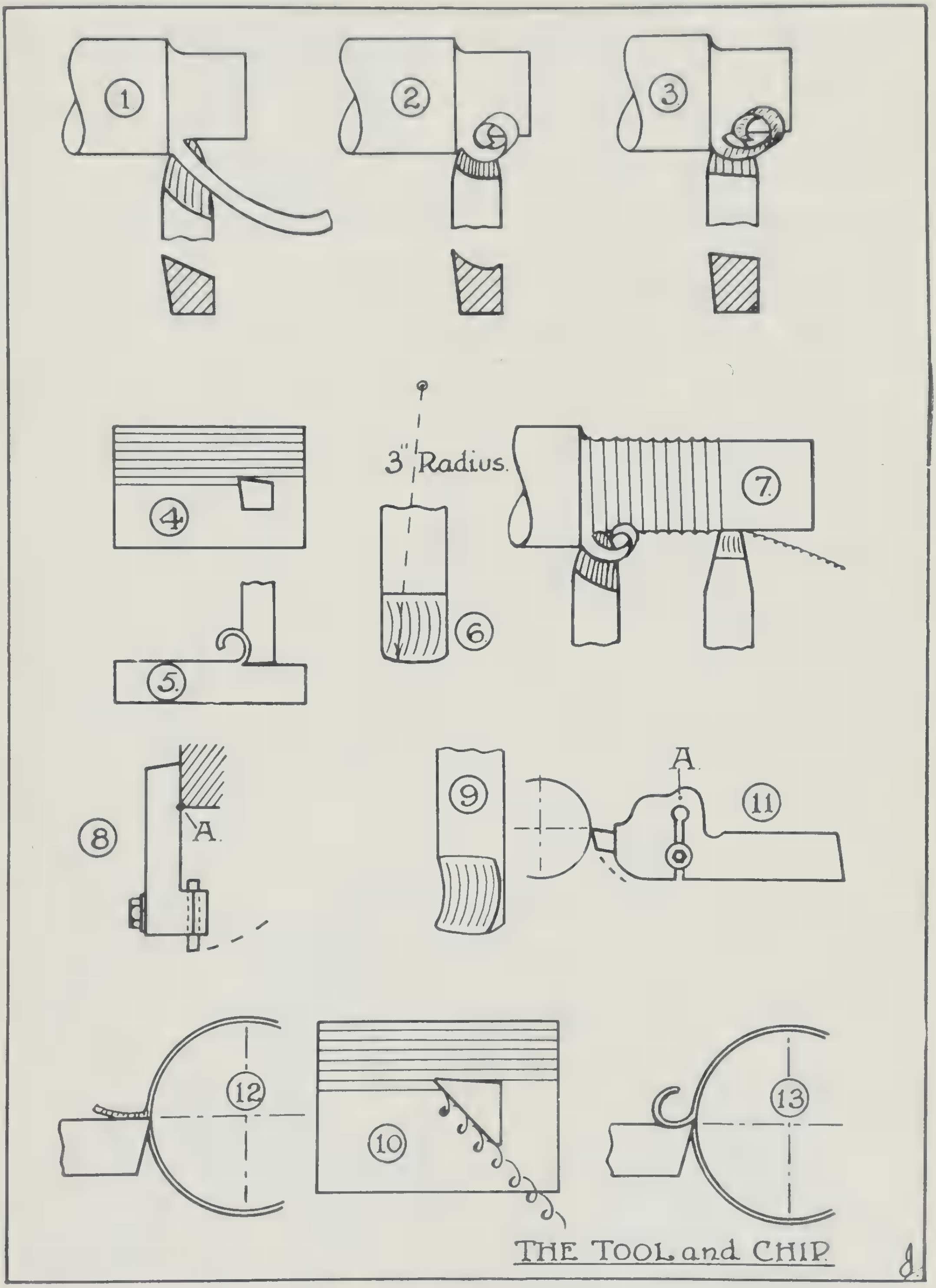
Diagram (1) shows the worst type of chip, a long stringy chip that comes away freely but in a long string and becomes entangled in the work. The reason for this is shown by the section of the tool taken along the line of the chip passage. The chip passes over a surface that offers free movement in its natural direction as it leaves the work.

Diagram (2) shows a tool with the lip shorter than in Diagram (1) and the section shows that the chip is forced to follow the curvature of the lip. The shape of the lip is therefore expressed in the chip and it is rolled up. If the cut is heavy, the effect of the forcing action of the lip shape exerts a rolling or breaking action on the chip sufficient to sever or crack it, and it breaks off in short curled sections.

Diagram (3) shows a tool with very little side rake and practically flat on the top. This tool will heat up much more than the one shown in diagram (2), because it offers more resistance to the chip. The resistance meets the chip just as it leaves the work, and the lip is so abrupt that it forces the chip upwards, and this tendency rolls the chip and breaks it into small sections.

Diagram (4) and (5) show the plan and elevation of a shaper tool, finish-planing cast iron. Diagram (6) shows an enlarged detail of the front face of the tool. This lip is curved slightly to give the chip an easy passage to reduce the chip pressure, although a light cut is taken. If this tool bit is used in a special planer tool holder, as shown in diagram (8), it will be noticed that the cutting edge is behind the last point of support of the tool holder at (A) and in consequence the tool springs away from the work and produces a smooth finish.

Diagram (7) shows a roughing tool and a finishing tool used on the lathe. The finishing tool has a comparatively broad nose in contact with the work and should be used with a coarse feed. There is great danger of


1
2
3
4
5
3" Radius.
6
7
8
A.
9
A.
11
12
10
13
THE TOOL and CHIP.

chatter with this type of tool because of the length of the contact between the cutting edge and the work. If the serrations caused by the roughing tool are only cut out with a light cut and the top face of the tool is lipped, smooth work will result. A toolholder similar to that shown in diagram (11) helps considerably in obtaining a smooth finish, because like the planer tool in diagram (8), it springs away from the work and does not dig in and cause chatter marks. The toolholder is designed on the goose neck principle and the point (A), being the weakest point, becomes the pivot point when the tool is under pressure at the cutting edge.

Diagrams (9) and (10) show the side view of a tool and the tool at work finish-planing wrought iron or steel. The tool works with a shear action and rolls the thin chip in a long closely packed shaving. The work is left bright and smooth, so different from the rough planing with an ordinary tool when the surface, due to its fibrous nature, is torn up, producing a rough serrated surface. (Use cutting compound).

Diagrams (12) and (13) show two parting tool blades at work. The flat top tool seems to break the chip unduly, giving considerable chip pressure, while the tool in (13) relieves the pressure by providing an easier passage for the chip and rolls the chip up and breaks it off in short coils. The drawback to lipping the tool in this way is that unless care is exercised, a negative rake will develop through careless lip-grinding, or the tool is ground down too low and wedges itself in the cut because the top edge is thicker than the cut made by the cutting edge when ground too low.

TOOLS FOR PRODUCTION MACHINES

This lesson introduces the various tools used in turret lathes and automatic screw machines. The same principles apply here as in previous lessons on tool grinding, but the setting of the tool in its relative position with the work plays an important part. It is very necessary to-day for a mechanic to understand how to tool up a machine so that parts may be produced by unskilled operators rapidly and be of good quality.

The position of the tool bit or blade. Diagram (1) shows a tangential blade and diagram (2) a radial blade similar to those used on an ordinary lathe. The tangential blade has the advantage over the radial blade because it is easier to adjust when reducing the diameter of work, and the body of the cutter is in direct support of the cutting edge as it meets the resistance of the chip.

In diagram (1) the end of the tool is ground to give the rake to the tool and the tool is inclined to give the side clearance, whereas in diagram (2) both side clearance and side rake must be ground on the tool.

Balance turning tools: Diagrams (3), (4) and (5) show three different views of a balance turning tool. The two blades are clamped by two set screws in a balance turning toolholder so that they operate on opposite sides of the work, each tool reducing the diameter of the stock in steps, as shown in diagram (3), while diagram (4) shows the share each tool is taking in reducing the diameter of the stock. Diagram (5) shows how the blades are inclined in the holder to give suitable side clearance, while the right angle side of the tool cuts to a square shoulder at the cutting edge.

Hollow mills are illustrated in Diagrams (6), (7) and (8). They are also balance turning tools, but have three cutting edges and are made of a solid piece of steel. Diagram (6) shows a hollow mill suitable for brass. It will be noticed that the cutting face has no rake, while the hollow mill in diagram (8) has a front rake and is used for reducing iron and steel.

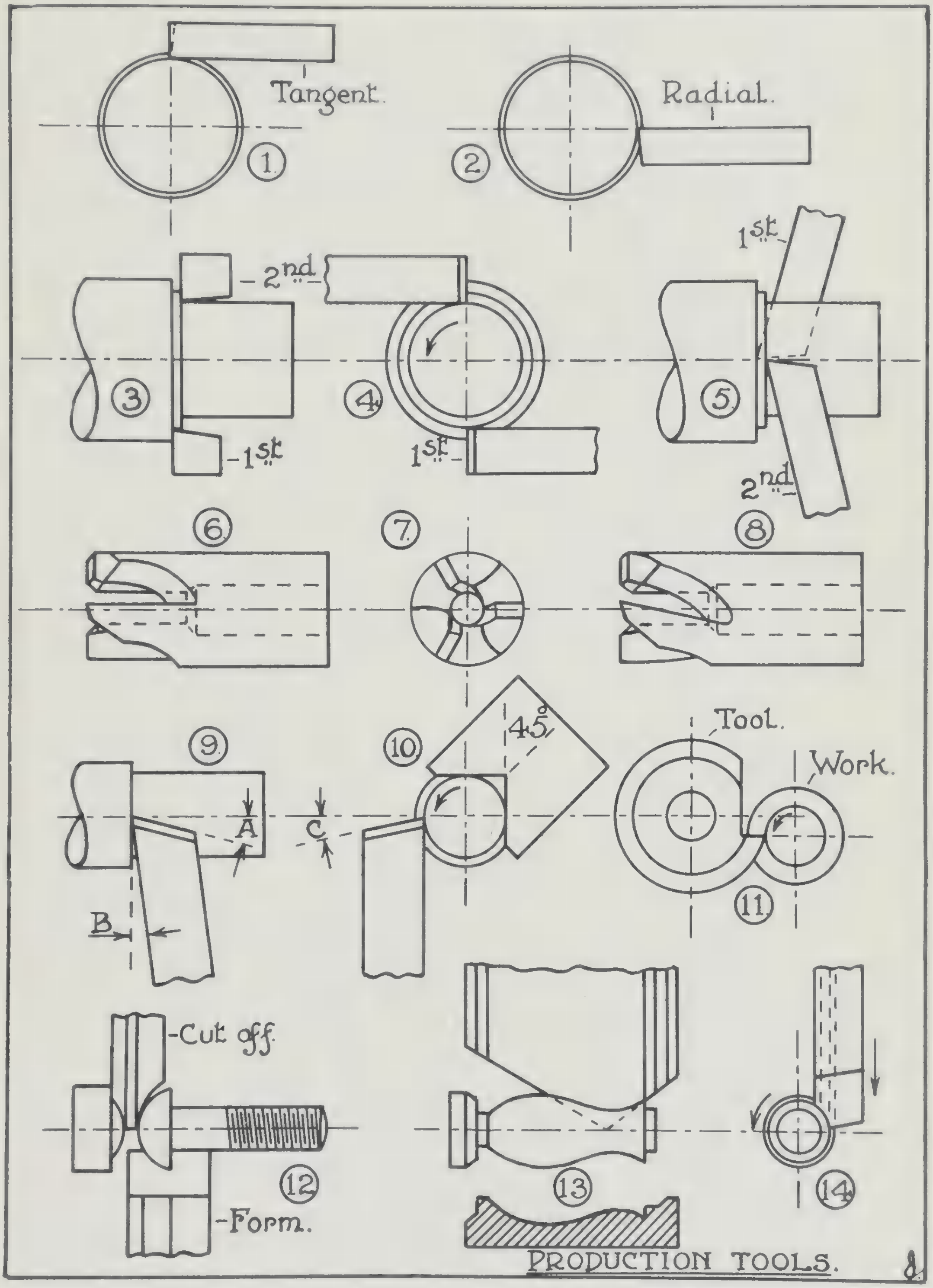
Hollow mills are used for rough turning only, to reduce rapidly the diameter of stock. They are very suitable for slender work, as the work when reduced passes through the mill and is supported while the cutting continues. The hole in a hollow mill is tapered, being slightly larger on the inside to prevent it rubbing the work. The cutting edges, it will be noticed by looking at diagram (7), are not radial, but are parallel to the radii and in advance of the centre line, as shown in diagram (8), so that as the edges are ground when being sharpened they will still cut effectively until the centre line is reached. It is preferable to have the

work bevelled slightly, a little smaller than the hole in the mill so that it will enter the mill and be cut true on centre. The cutting edges on the inside of the mill are bevelled at 45° to receive the work.

Cutting tool with back rest: V type and roller back rests are used to support the work while being turned. They bear against the finished surface of the work and therefore follow the tool as it cuts. Both sides of the V rest should be hard and smooth and should bear with equal pressure on the work to prevent the work from being forced out of line. The angle (A) of the tool in diagram (9) gives the side rake, angle (B) the side clearance, and angle (C) the front rake.

Circular forming tools of a type shown in Diagram (11) may be used to form work to required shape or for cutting off purposes. The form required on the work is cut on a circular disc of steel and a notch is cut into it, as shown, to provide a cutting edge and can be ground to sharpen the edge without changing the form. Diagram (12) shows a screw being formed by a circular forming tool and being finish-formed and cut off by the circular forming and cutting off tool. Forming tools of this type are held by a cap screw through the centre of the disc while a hook bolt clamps the disc to the holder to prevent it from turning under the chip pressure.

The skiving tool is illustrated at work forming a piece of stock in diagrams (13 and 14). It is a tool that is rarely used and is intended for forming brass or other soft material, where a wide form is required that runs down to a small diameter, making the work comparatively weak to resist the cutting action of the tool. The top face of the cutter is formed and passes the rotating metal, as shown, leaving it the same form as itself. The tool is pointed at the end so that the cutting action may be gradual. The last point of support near the chuck is cut last so that it is strong enough to support the metal while the part away from the chuck is being formed.



Tangent.
1
Radial.
2
2nd
1st
3
4
1st
5
2nd
1st
6
7
8
9
A
B
C
10
45°
Tool.
Work.
11
-Cut off.
12
-Form.
13
14
PRODUCTION TOOLS.

GRINDING

1. Name 3 natural abrasives. What defects do natural abrasives often possess?
2. What do at least 2 natural abrasives consist of?
3. What are the 2 main classes of artificial abrasives? What materials are each suitable for grinding?
4. Give at least 2 trade names for each of 2 artificial abrasives.
5. What is each of 2 artificial abrasives made from? Where are the materials found?
6. What is the function of a bond? What bonds are commonly used?
7. What are the 4 chief processes of wheel manufacture?
8. Why is Porosity in a grinding wheel necessary?
9. What is it that varies the grade of a grinding wheel? How is it tested?
10. Give details of any manufacturer's Grade Scale.
11. How are the grain sizes obtained? How do you select the grain size for certain kinds of work?
12. Give the particular features of wheels made by at least 3 distinct processes of wheel manufacture.
13. Give approximate ratings of the following grain sizes: (a) Very coarse, (b) Coarse, (c) Medium, (d) Fine, (e) Very fine.
14. Give approximate ratings of the following grades, (a) Norton, (b) Carborundum; (1) Soft, (2) Very soft, (3) Medium, (4) Hard, (5) Very hard.
15. Sketch 3 views of a tool bit, and show the following angles. Front and side clearance and side cutting angle. Front and side rake and front cutting angle. Profile.
16. Sketch the profile of an ideal tool for rough turning with the following advantages: (a) Small amount of metal removed when ground, (b) Rigid, (c) Will dissipate heat from the cutting edge, (d) Will leave the surface of the work smooth.
17. Illustrate the profile of a lathe tool cutting the rough skin from Cast Iron.
18. Which tool receives the greatest chip pressure, (a) A round-nosed profile, (b) A straight-edged profile? Depth of cut and feed being equal.
19. Sketch the profile of a tool for cutting (a) Slender work, (b) Heavy work. Show the effect of the forces acting on the tools.
20. What is it that governs the amount of side clearance a lathe tool should have?
21. Illustrate with a sketch the end view of a piece of work being turned, showing the height of the tool and the proper front clearance.
22. If a Shaper tool had too much clearance what effect would it have on the tool, and on the work?

23. When is front rake necessary and when unnecessary?
24. What is the effect of chip pressure on (a) a negative and (b) a positive front rake?
25. What is the effect of: (a) Too much side rake? (b) Not enough side rake?
26. State the advantages and disadvantages of: (a) a solid forged tool, (b) a tool bit and toolholder.
27. Sketch a common type of toolholder showing the angles on the tool bit as usually supplied.
28. Sketch a solid tool for turning: (a) Soft Steel, (b) Hard Steel, (c) Brass.
29. Sketch a tool bit in its regular position as held in a toolholder for turning: (a) Chilled Iron, (b) Cast Iron, (c) Medium Hard Steel.
30. Illustrate by means of an enlarged sketch a tool removing a chip from a piece of work in the lathe. Show the important features of the separation of the metal from the work.
31. Why should the edge of a tool be stoned to a keen cutting edge?
32. Which tool cuts work in the lathe with the least resistance to the chip; a straight topped tool, or a lipped tool? State reasons why.
33. Sketch a chip from: (a) Soft Steel, (b) Hard Steel, showing tool and work.
34. Draw the profile of a tool suitable for turning brass.
35. Sketch 3 views of a parting tool ground from a tool bit, show its position in relation to work on the lathe.
36. Sketch an enlarged profile of a tool suitable for turning work smooth to a shoulder.
37. Sketch 2 views of spotting tool.
38. If a drill is to follow a spotting tool, sketch in section the conic recess made by the spotting tool, and the drill point in position.
39. What influences are brought to bear on a boring tool set in the following positions: (a) above centre, (b) on centre, (c) below centre? Which is correct?
40. Sketch lathe tools turning machine steel: (a) One that produces long stringy chips, (b) One that produces continuous curly chips, (c) One that breaks up the chips into short lengths.
41. Sketch a finish tool: (a) For turning Cast Iron, (b) For planing Cast Iron, (c) For planing Steel.
42. Sketch a goose neck or spring tool for: (a) A Shaper, (b) Lathe.
43. Sketch a tangent tool and a radial tool used for turning on turret lathes.
44. Sketch an end view of a circular forming tool and show its relation to the work being turned.
45. Sketch a turning tool in operation on work held against a back rest.
46. Make a sketch in section of a hollow mill reducing a bar of stock in the lathe.
47. When is a skiving tool used in forming work in the lathe?

SHOP MATHEMATICS

GEAR RATIOS

Gear ratio. Diagram (1) shows a gear driving a pinion. The effect of this drive will be two-fold.

(1) It will change the direction of rotation because there are only two gears in mesh.

(2) It will change the number of revolutions per minute, in proportion to the number of teeth in the mating gears.

If the gear has 90 teeth and the pinion 30 teeth, it will be necessary for the pinion to rotate 3 times before the gear makes one complete revolution.

The gear ratio is the ratio of the number of teeth in the mating gears and governs the relative rotation of the driver and follower. Since the mating gears must have the same diametral pitch, it is obvious that their diameters are also proportional in the same ratio as the number of the teeth.

Example: Let S = speed of driver.
and s = speed of follower.
Let N = number of teeth in driver.
and n = number of teeth in follower.

Then S X N = s x n.

If the driver gear with 90 teeth makes 100 R.P.M. Find speed of follower gear with 30 teeth.

Solution: Speed of driver X number of teeth = speed of follower X number of teeth, or S X N = s x n.

i.e. $100 \times 90 = s \times 30$. Therefore $s = \dfrac{100 \times \overset{3}{\cancel{90}}}{\cancel{30}}$

s = 300 R.P.M.

The speed of the driver and follower is inversely proportional to the number of teeth in the gears, so that since the ratio is 3 to 1 the follower will go 3 times as fast as the driver—3 X 100 = 300 R.P.M.

Diagram (2) shows a simple train of gears.

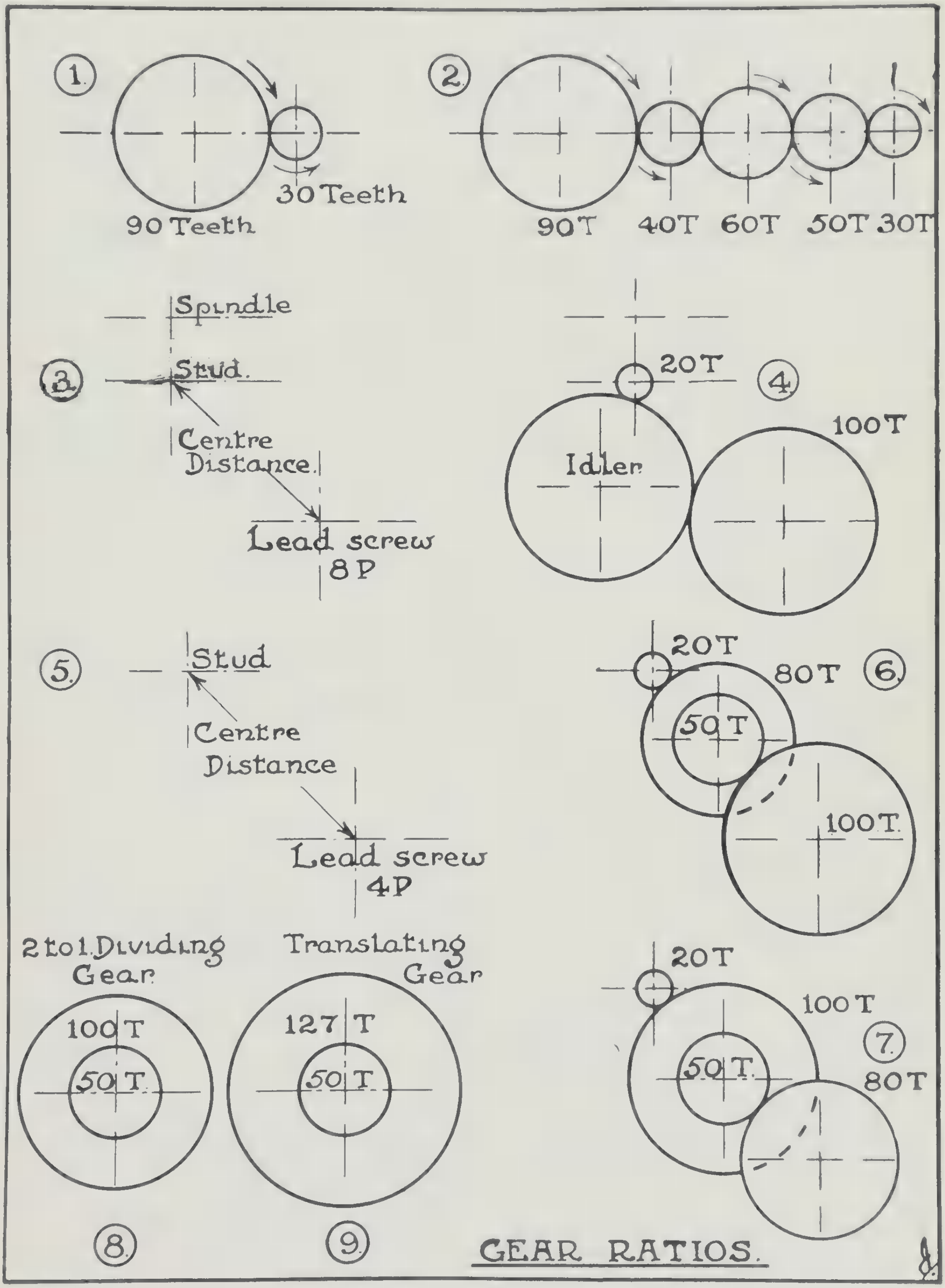
What is the influence of the idler gears?

Speed of driver X number of teeth in all drivers = speed of follower X number of teeth in all followers.

There are 4 drivers and 4 followers. Therefore S X N X N X N X N = s x n x n x n x n.

If the speed of the first driver is 100 R.P.M. what is the speed of the last follower? $100 \times 90 \times \cancel{40} \times \cancel{60} \times \cancel{50} = s \times 30 \times \cancel{50} \times \cancel{60} \times \cancel{40}$

$$s = \frac{100 \times \overset{3}{\cancel{90}}}{\cancel{30}} \qquad \underline{s = 300 \text{ R.P.M.}}$$


1.
90 Teeth
30 Teeth
2.
90T
40T
60T
50T
30T
3.
Spindle
Stud.
Centre Distance.
Lead screw 8 P
4.
20T
Idler
100T
5.
Stud
Centre Distance
Lead screw 4P
6.
20T
80T
50 T
100 T.
2 to 1. Dividing Gear
100 T
50 T.
Translating Gear
127 T
50 T
7.
20T
100 T
50 T.
80T
8.
9.
GEAR RATIOS.

From the above equation it can be seen that the idlers do not affect the ratio. Their only function is to connect the gears, and since they make an odd number in the train they do not change the direction of rotation of driver and follower.

Simple train on lathe. The ratio of a simple train on a lathe is usually not greater than 1 to 5, because the centre distance as shown in diagram (3) limits the ratio.

Note. We will assume in this lesson that the spindle and stud have a ratio of 1 to 1.

How many threads will be cut on work between centres if the lead screw is 8 P. and the lathe is geared as shown in diagram (4). Threads on work X driver = Threads on lead screw X follower.

$$\text{Therefore } T \times 20 = 8 \times 100, \qquad T = \frac{8 \times \overset{5}{\cancel{100}}}{\cancel{20}} \quad \therefore \quad \underline{T = 40.}$$

Note. The idler only connects the gears. Its effect on the rotation of the lead screw can be changed by the reverse gears. For right hand threads the tool moves towards the live spindle. For left hand threads away from the spindle.

Compound train on lathe. When the ratio of threads on lead screw to threads on work is greater than 1 to 5, a compound train must be used. (Diagram 6).

Diagrams (5) and (6). What number of threads will be cut on work between centres if the lead screw is 4 pitch and the lathe is geared as shown? $T \times 20 \times 50 = 4 \times 100 \times 80$.

$$\text{Therefore } T = \frac{4 \times \overset{2}{\cancel{100}} \times \overset{4}{\cancel{80}}}{\cancel{20} \times \cancel{50}} \qquad \therefore \qquad \underline{T = 32}$$

A dividing gear, diagram (8), is a double gear used in a compound train and has usually 100 teeth and 50 teeth, or a ratio of 2 to 1. It is obvious if this is used as in diagram (7) the final ratio will be double that of the first and last gear. 20 to 80 = 1 to 4, with dividing gear this becomes 1 to 8, so that if there are 4 threads on the lead screw, the threads on the work will be 4 x 8 = 32 threads.

A translating gear is a double gear with 50 and 127 teeth, which is the relation between 1 in. and 1 cm. It is used on a lathe with a lead screw with so many threads per in. to produce so many threads per cm. on work between centres according to the ratio of the other gears in the train.

GEARING FOR THREADS

A lathe operator should understand how to gear up his lathe to cut threads. Lathes are usually provided with a chart or index plate, showing the gears to use for cutting threads, and this chart gives the operator a list of the gear numbers provided with the machine. It is well to note what the gear increment, progression, or constant is, as some lathes have a 4 increment and some a 5. This is not rigidly followed, because by varying the numbers slightly, greater combinations may be made with a minimum number of gears.

Question 1. Work out suitable gears for a lathe with an 8 p (8 pitch) or (8 threads per inch) lead screw, to cut 18 threads per inch on work between centres. See diagrams (1), (2) and (3).

Note. Spindle and stud ratio 1 to 1, 18 threads on work and 8 threads on lead screw.

Ratio 9 to 4 (divide 18 and 8 by 2).

Multiply each by some number to make the lowest number not less than 24 because it is the smallest gear in the set provided.

$$\left.\begin{array}{l} 4 \times 6 = 24 \\ 9 \times 6 = 54 \end{array}\right) \text{The two gears required.}$$

Now where shall the gears be placed?

To cut finer threads than the lead screw, the lead screw must turn slower than the spindle, so that the large gear goes on the lead screw and the smaller one on the stud. This ratio of 9 to 4 being less than 1 to 5 is a simple train, so an idler is used to connect them, see diagram (1).

Question 2. Work out a suitable train of gears to cut 32 thread per inch on work between centres, when the lead screw is 6 P.

Note: Ratio 6 to 32 is greater than 1 to 5, therefore a compound train would be used.

Method (1). 32 threads on work and 6 threads on lead screw.
Ratio 6 to 32 or 3 to 16.

Factorize. $6 = (3) \times (2)$
and $32 = (8) \times (4)$

Multiply each pair by some number to give numbers that may be found in the set without duplication, as there is usually only one pair of gears of the same number available in a set.

1st pair $\left(\begin{array}{l} 3 \times 8 = 24 \\ 8 \times 8 = 64 \end{array}\right)$ **2nd pair** $\left(\begin{array}{l} 2 \times 20 = 40 \\ 4 \times 20 = 80 \end{array}\right)$

Because the threads on the work are finer than those on the lead screw, the small number of each pair will be the driver and the larger number of each pair the follower.

To apply to Diagram (4):

A = 24 teeth—driver.
B = 64 " —follower.
C = 40 " —driver.
D = 80 " —follower.

To prove work. Threads on work x all drivers, equals threads on lead screw x all followers.

$$T \times \overset{4}{\cancel{24}} \times \cancel{40} \times = \cancel{6} \times \overset{2}{\cancel{80}} \times \overset{16}{\cancel{64}}.$$

$T = 2 \times 16.$
$T = 32.$

Method 2. Rapid method by using a 2 to 1, Dividing gear in the compound train.

Example: 32 threads on work.
6 " " lead screw.

The Dividing gear cuts this 6 to 32 ratio in half so that it now becomes 6 to 16.

$6 \times 4 = 24$ first driver.
$16 \times 4 = 64$ last follower.

Applying the gears to Diagram (4)

$$\begin{array}{l} A = 24 \text{ teeth.} \\ \left.\begin{array}{l} B = 100 \text{ "} \\ C = 50 \text{ "} \end{array}\right\} \text{ Dividing gear.} \\ D = 64 \text{ "} \end{array}$$

Use of idler in compound train as shown in Diagram (4). This becomes necessary when the gear on the lead screw is so small that it could not engage with the small gear on the adjustable stud, without the large gear on the adjustable stud rubbing the lead screw.

Gearing up a lathe when the spindle and stud gears are not a ratio of 1 to 1. (See Diagram (7). Before gearing a lathe of this kind, it is necessary to know the lead number of the machine.

Lead number. Is the number of threads that can be cut on the machine with equal gears on stud and lead screw. In diagram (7) the lead number of the lathe would be $2/1 \times 4 = 8$, the lead number because it takes 2 turns of the spindle, before the lead screw turns once if it is equally geared on stud and lead screws.

Question 3. Find the gears to cut 24 thds. per inch on work if the lead screw is 4 pitch and the lead of the machine is 8. (Gear constant 4).

Ratio of threads on work to lead of machine 24 to 8 = 3 to 1. Multiply 3 and 1 by a suitable number which is a multiple of 4 to give gears less than 100 teeth

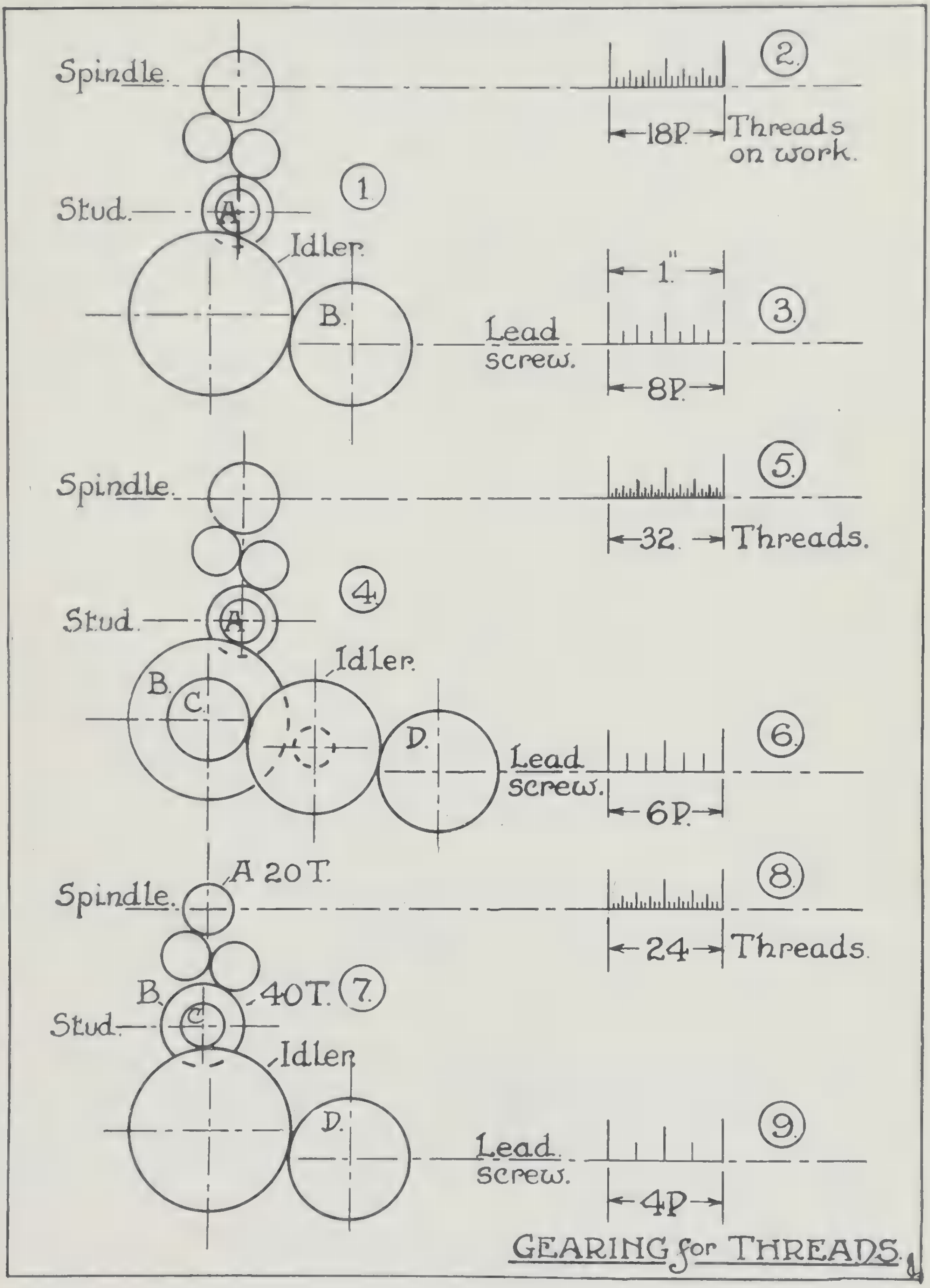
$3 \times 24 = 72$ Follower D (diagram 7).
$1 \times 24 = 24$ Driver C (diagram 7).

Proof:—(Start at spindle, not stud).
Threads on work x A x C = Threads on lead screw x D x B.

$$T \times \cancel{20} \times \cancel{24} = 4 \times \cancel{72} \times \cancel{40}$$

$$\therefore \quad T = \frac{4 \times \overset{3}{\cancel{72}} \times \overset{2}{\cancel{40}}}{\cancel{20} \times \cancel{24}}$$

$$T = 24$$


Spindle.
Stud.
A
Idler.
B.
1
Lead screw.
2
18P.
Threads on work.
1"
3
8P.
Spindle.
Stud.
A
4
Idler.
B.
C.
D.
Lead screw.
5
32.
Threads.
6
6P.
A 20T.
Spindle.
B.
40T.
7
Stud.
C.
Idler.
D.
Lead screw.
8
24
Threads.
9
4P.
GEARING for THREADS.

PROBLEMS ON THE SQUARE AND CIRCLE

Question. Diagram (1):

What is the largest square shank that can be milled on a screw or tap 1¼″ diameter?

Rule: Size of square = Diameter x .7071.
" " " = 1.250″x .7071.
" " " = .884″.

Question. Diagram (2):

If a hole has to be drilled to receive a nut, what is the distance across the corners of the nut if the distance across the flats is 1½″ (square nut for ¾″ bolt)?

Rule: The diagonal of a square equals the side of a square multiplied by 1.414.

Distance across the corners when 1½″ across the flats = 1.5″ x 1.414 = 2.121″ across corners.

Question. Diagram (3):

What must be the diameter of round stock so that a square bolt head 2″ across flats may be machined from it?

Rule. Diameter of round stock $= \frac{\text{size of square.}}{.7071}$

" " " " $= \frac{2''}{.7071}$

" " " " $= 2.828''$

Question. Diagrams (4), (5) and (6).

What is the bore of a single cast iron pipe to equal the total area of two cast iron pipes 8″ and 6″ bore?

Take a steel square, as in diagram (6), and if 8″ is marked off on one blade and 6″ on the other and then the hypotenuse of the triangle is measured, it will be found to be 10″. This distance is the diameter of the bore required to equal the area of the two pipes.

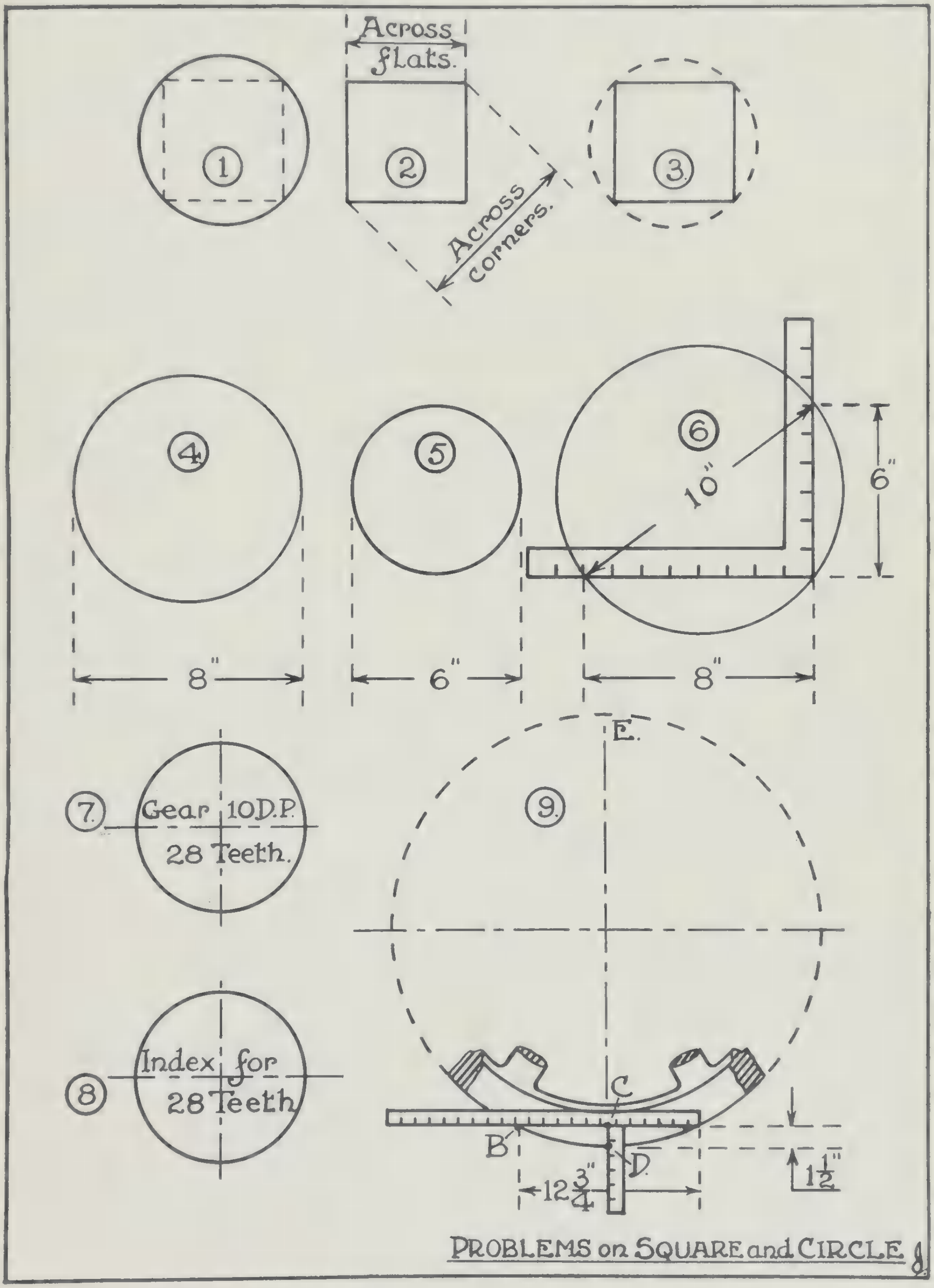
Rule: In a right-angled triangle the square on the hypotenuse equals the square on the other two sides. Similarly, circles whose diameters are in the same ratio as the sides of a right-angled triangle will have areas in proportion.

Question. Diagram (7):

What will the outside diameter of a gear blank be, when turned for a 10-pitch gear with 28 teeth?

Rule: Divide the number of teeth plus 2 by the diametral pitch.

Outside diameter $= \frac{28+2.}{10}$ Outside diameter = 3″.


Across
flats.
Across
corners.
1
2
3
4
5
6
10"
6"
8"
6"
8"
E.
7
Gear 10D.P.
28 Teeth.
9.
8
Index for
28 Teeth
C
B
D.
12 3/4"
1 1/2"
PROBLEMS on SQUARE and CIRCLE

Question. Diagram (8):

What index will be used on the index head of a milling machine to cut 28 teeth on a gear blank?

Note: Brown and Sharpe plates have numbers

Set (1) 15, 16, 17, 18, 19, 20,
" (2) 21, 23, 27, 29, 31, 33,
" (3) 37, 39, 41, 43, 47, 49.

Ratio of worm and worm gear in index head 40 to 1.

Rule: Divide 40 by the number of teeth in the gear to be cut.

$$\text{Index} = \frac{40}{28} = 1\frac{12}{28} = 1\frac{3}{7}$$

this is 1 turn of crank and $\frac{3}{7}$ of a turn.

Select holes in plate a multiple of 7. Holes in plate selected = 49.

Index = 1 turn and 21 holes in a 49 circle.

$$\frac{3 \times 7}{7 \times 7} = \frac{21}{49}$$

$$\text{If a 21 circle is selected Index} = \frac{3 \times 3}{7 \times 3} = \frac{9}{21}$$

Index = 1 turn and 9 holes in a 21 circle.

Note: If holes in plate cannot be found, gearing must be used in addition to the index plate. This is known as differential indexing.

Question. Diagram (9):

Find the outside diameter of a gear wheel, when only a broken section of the gear wheel is available.

Rule: In a circle CD:CB as CB:CE. Apply this to the problem given. CE x 1.5″ = 6.375″ x 6.375″.

$$\text{Therefore CE} = \frac{6.375'' \text{ x } 6.375''.}{1.5''}$$

Therefore CE = 27.093″.

Outside Diameter = CE + CD.
" " = 27.093″ + 1.5″.
" " = 28.593″.

INTRODUCTION TO APPLIED TRIGONOMETRY

The elementary principles of trigonometry will be found to be of great help to a workman when applied to practical problems. They will permit him to work to a great degree of accuracy and assist him to work out a speedy solution to a problem. Trigonometry treats of the properties of triangles, and enables one to find the dimensions of all the angles and sides of a triangle when those of only some are known.

Experiment: If a hinged rule were taken, as in diagrams (1), (2) and (3), and the side AB gradually rotated about the hinge A, while AD remains horizontal and BC vertical, it will readily be observed that as the angle B A D increases so B E increases in length and A E decreases in length, at the same time the angle ABC decreases in size. This experiment proves conclusively that there is a definite relation between the sides of the triangle and the angles of the triangle.

There are several functions that determine the value of the respective angles and the lines by which the angles are formed. The most important functions are shown in diagram (4). They are the sine, cosine, tangent and co-tangent.

In reference to the table of functions used in trigonometry it must be understood that the values given are based on angles calculated in a circle 1″ radius.

In diagram (4) BC is the sine of the angle 60° and equals .86603″. AC is the cosine of the angle and equals .5″. DE is the tangent of the angle and equals 1.73205″. FG is the cotangent of the angle and equals .57735″. These values are constant for 1″ radius and a 60° angle.

Diagrams (5) and (6) show the names of the sides used for a given angle. The long side is always the hypotenuse. The side next to the angle is the side adjacent, and the third side is the side opposite. The following formulas can then be supplied:

(1) $\text{Sine} = \dfrac{\text{side opp.}}{\text{hyp.}}$

(2) $\text{Cosine} = \dfrac{\text{side adj.}}{\text{Hyp.}}$

(3) $\text{Tangent} = \dfrac{\text{side opp.}}{\text{Adj.}}$

(4) $\text{Cotangent} = \dfrac{\text{side adj.}}{\text{side opp.}}$

(5) $\text{Hyp.} = \dfrac{\text{side adj.}}{\text{Cosine}}$ or $\text{Hyp.} = \dfrac{\text{side opp.}}{\text{Sine.}}$

Question. Three holes have to be located in a die block, as shown in diagram (7). What is the exact centre distance between the other holes?

The centre lines of the holes form a right-angled triangle ABC.

Solution for distance BC would be found from the tangent for a 25 degree angle; **this is .46631.**

side opp. = side adj. x tangent.
BC = 2.25″ x .46631 = 1.049″.

For the length AC, the solution is as follows:

$$\text{Hyp. AC} = \frac{\text{side adj.}}{\text{cosine.}}$$

$$\text{Therefore AC} = \frac{2.25''}{.90631} = 2.4826''.$$

Question. Twenty holes are required to be equally spaced on a circle 18″ in diameter. What distance must the divider points be set to step off the hole centres? The enlarged drawing, diagram (8), shows the two radii AO, and BO and the chord AB, which it is required to find. As there are 20 holes in the circle, the angle formed by the two radii OA and OB $= \frac{360°}{20} = 18$ degrees.

Therefore angle COB = 9 degrees.

In the right-angled triangle OCB the angle COB and hyp. BO are known.

Find CB:—Side opp. CB = Hyp. x sine. CB = 9″ x .1564 = 1.4076″.

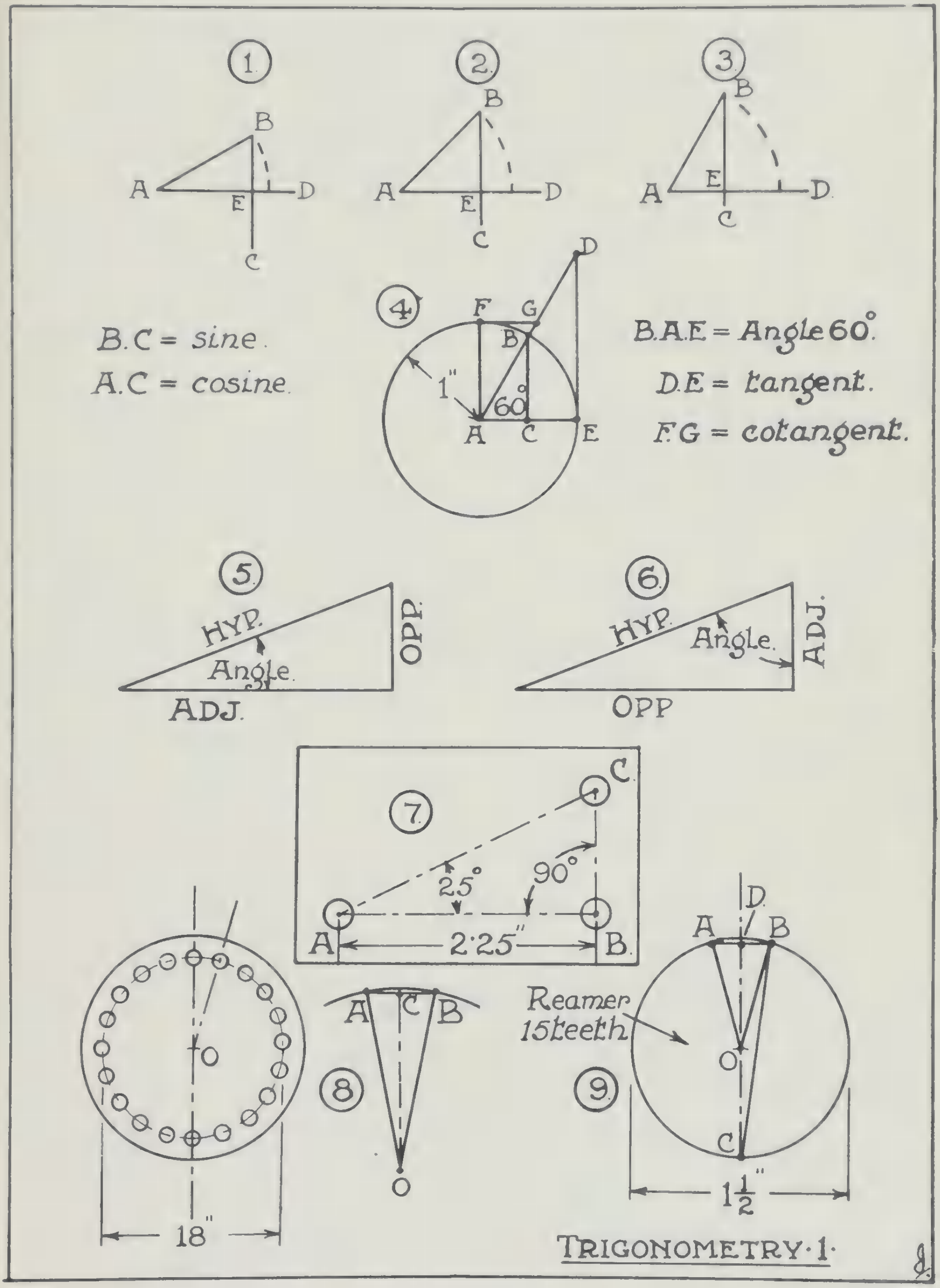
Set divider 2 x 1.4076″ = 2.8152″.

Question: Find the distance across the two cutting edges nearest to the centre of a 1½″ reamer with 15 teeth. See diagram (9).

A and B represent two cutting edges of the reamer opposite a cutting edge C. The centre angle AOB $= \frac{360°}{15} = 24°$. To find length of side opp. DB, or ½ the chord AB equals the sine of ½ centre angle multiplied by radius. DB = sine of 12° or .2079 x .75″ = .1559″.

The chord BC $\frac{\text{side opp.}}{\text{sine}} = \frac{.1559''}{\text{sine } 6°} == \frac{.1559''}{.1045}$ The chord BC = 1.492″.

This is .008″ less than if the teeth where diametrically opposite.


B.C = sine.
A.C = cosine.
B.A.E = Angle 60°.
D.E = tangent.
F.G = cotangent.
HYP.
Angle.
OPP.
ADJ.
25°
90°
2·25"
Reamer 15 teeth
18"
1½"
TRIGONOMETRY·1·

PROBLEMS IN TRIGONOMETRY

Problem Diagram (1). A piece of machine steel must be turned 4″ long, 1″ diameter at the small end with a taper of 5° with the axis of the work. What will the diameter of the large end be?

In the right-angled triangle ABO the side AO must be found. The adjacent side AB is given, 4″. The angle ABO is given 5°.

$\text{Tan} = \dfrac{\text{side opp.}}{\text{side adj.}}$ Therefore side opp. = Tan x side adj.

Tan of 5° = .08749. Therefore side opp. = .08749 x 4″ = .34996″. Dia. D.O. = 1″ + 2 x .34996″ = 1.69992″.

Problem Diagram (2). To find the diameter across the sides of a regular polygon. A screw 2½″ diameter must have 6 equal sides milled on its end. What will be the distance across the flats?

The angle AOB $= \dfrac{360°}{6} = 60°$. Therefore the angle COB = 30°.

In the right-angled triangle COB, the angle COB = 30°. BO = 1.250″, Cos. of 30° = .86603. $\text{Cos.} = \dfrac{\text{side adj.}}{\text{hyp.}}$ Therefore side adjacent = hyp. x cosine.

Side adj. OC = 1.250″ x .86603 = 1.08253″.

Diameter = 2 x 1.08253″ = 2.16506″.

Rule. To obtain the diameter across the sides of a regular polygon when the diameter across the corners is given. Multiply the cosine of the angle obtained by 360° divided by twice the number of sides, by the diameter across the corners, and the product will be the diameter across the sides.

Application to problem diagram (2).

Diameter across sides = cos. $\dfrac{360°}{12}$ x 2.5″ = cos. 30° x 2.5″ = .86603 2.5″ = 2.1650″.

Problem Diagram (3). A piece of steel must be milled with a hexagonal end 1½″ across the flats or sides. What diameter must the piece be turned to?

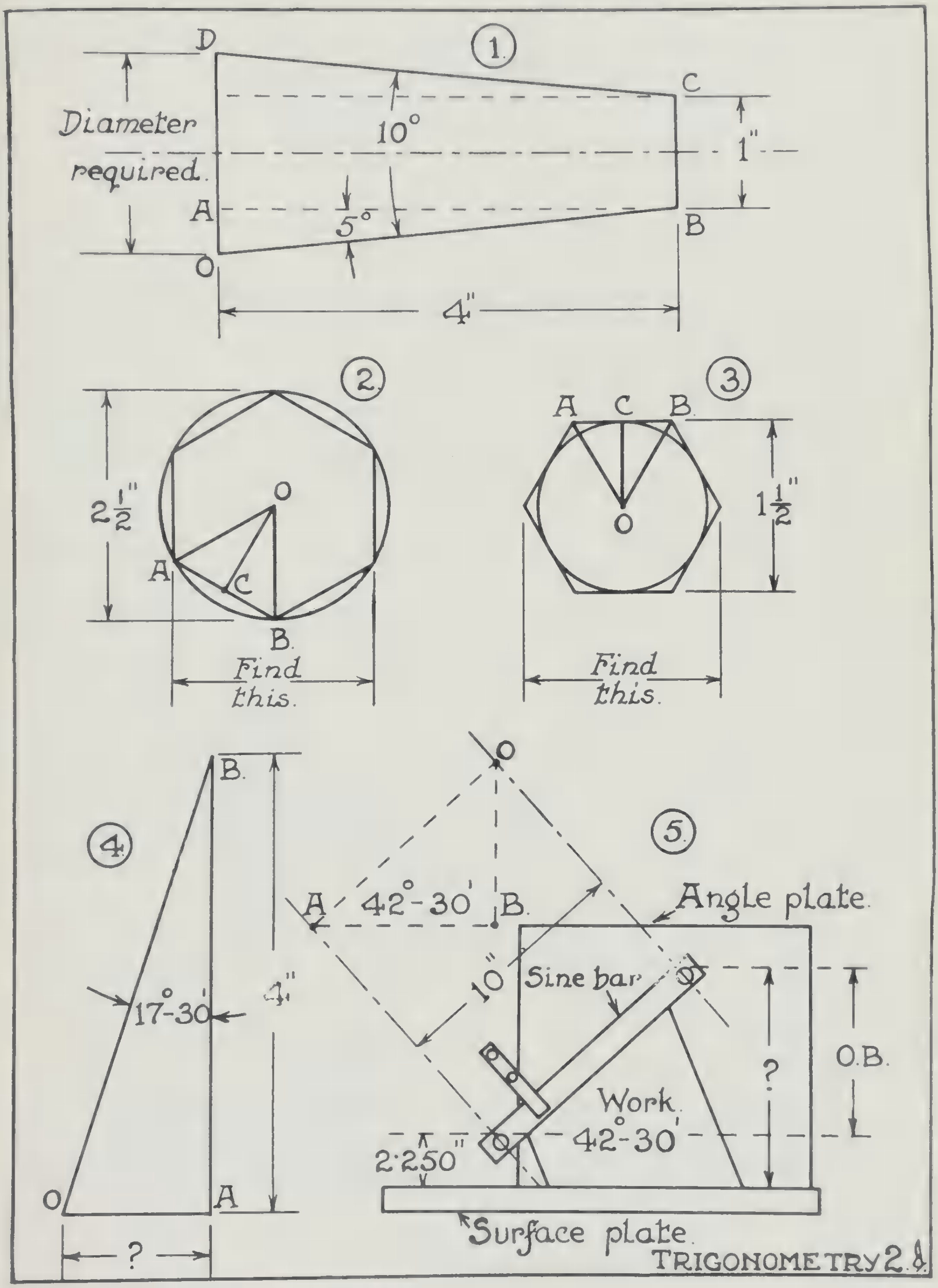
The angle AOB $= \dfrac{360°}{6}$ Therefore the angle COB = 30°.

In the right-angled triangle COB, angle COB = 30° and the side adj. CO $= \dfrac{1.5''}{2} = .750''$. Cosine of 30° = .86603. $\text{Cosine} = \dfrac{\text{side adj.}}{\text{hyp.}}$

Therefore hyp. $= \dfrac{\text{side adj.}}{\text{cos.}}$ hyp. $= \dfrac{.750''}{.86603} = .8660''$.

Diameter = 2 x .8660″ = 1.7320″.

Rule. In a regular polygon, when the diameter across the sides and the number of sides are given, to find the diameter across the


1
D
C
Diameter
required.
10°
1"
A
5°
B
O
4"
2
3
A C B.
$2\frac{1}{2}$"
O
$1\frac{1}{2}$"
A
C
B.
Find
this.
Find
this.
O
B.
4
5
A 42°-30' B.
Angle plate.
10"
Sine bar
4"
17°-30'
?
O.B.
Work.
42°-30'
2·250"
O
A
?
Surface plate.
TRIGONOMETRY 2.

corners, divide the distance across the sides by the cosine of (180° divided by the number of sides) and the quotient will be the distance across the corners.

Application to problem in Diagram (3).

$$\text{Distance across corners} = \frac{1.5''}{\cos.\ \frac{180^\circ}{6}} = \frac{1.5''}{\cos.\ 30^\circ}.$$

$$\text{Distance across corners} = \frac{1.5''}{.86603} = 1.7320''.$$

Problem Diagram (4). An angle of 17° 30′ is required for a template. The side opp. OA is therefore to be found. The tan of 17° 30′ = .31530″ when AB is 1″. Enlarge to 4 times the size to get more accurate results.

Multiply the side adj. AB and side opp. OA by 4.

side adj. = 1″ x 4 = 4″. side opp. = .31530″ x 4 = 1.2612″.

To lay out angle. Draw AB 4″ long and OA at right angles to AB 1.262″ long. Join OB, then angle OBA = 17° 30′.

The use of the Sine Bar. Diagram (5). A sine bar is an accurately made instrument consisting of a ground bar with two ground pins or plugs protruding from one side. It is used for measuring angles accurately and for locating work accurately to a precise angle. The pins or plugs are of the same diameter and are located in parallel alignment to the edge of the bar. The centre to centre distance of the plugs is usually 10″ for simplicity in figuring. The dotted right-angled triangle shows that the hypotenuse AO = 10″ and the side opp. OB = the difference in height of the two plugs. To set sine bar for an angle of 42° 30′ find length of side opp. which is OB. Sine of 42° 30′ = .67559. Now multiply the sine by 10″, the distance between the plug centres, 10″ x .67559 = 6.7559″. Adjust the sine bar with a vernier height gauge so that if the lower plug is 2.250″ high, the top plug will be 2.250″ + 6.7559″=9.0059″ high. The tool is now ready to test work for an angle of 42° 30′.

To measure angle of work. Set the sine bar accurately to fit the work. Measure the height of each plug and find the difference in the height. Example: *Difference* in height 6.7559″. Divide 6.7559″ by 10 = .67559″. The angle for .67559 is found from the tables of sines to be 42° 30′.

PROBLEMS IN APPLIED TRIGONOMETRY

Problem Diagram (1): A piece of tool steel must be turned to the dimensions given, while being held in the chuck of a lathe. What angle will the compound slide rest be set at with the lathe alignment?

The angle ACB in the right-angled triangle, ABC is required. This is the angle the outside of the work makes with the axis of the work.

Given side adj. $= 2.5''$. Side opp. $= \dfrac{.5'' - .125''}{2} = .1875''$.

$$\text{Tan} = \frac{\text{side opp.}}{\text{side adj.}} = \frac{.1875''}{2.5''} = .075.$$

From the tables of tangents the angle for .075 = 4° 17′. The compound rest is set at 4° 17′ with the lathe alignment, to produce the work in diagram (1).

Problem Diagram (2): A socket must be bored out on a lathe to the dimensions given. What is the angle ABC so that a taper attachment can be used to give the required taper, if BC = 3″?

In the triangle ABC side adj. $= 3''$ and side opp. AC $= \dfrac{.725'' - .600''}{2}$ $= .0625''$. Tan $= \dfrac{\text{side opp.}}{\text{side adj.}} = \dfrac{.0625''}{3''} = .02083$.

From tables of tangents the angle for .02083 = 1° 12′. This angle will be set on the angle graduation on the taper attachment.

Problem Diagram (3): A piece of cast iron must be recessed while being held in the chuck of a lathe. From the given drawing find the diameter D so that the angle may be checked by measurement. The beveled face of the recess forms the hyp. of a right angled triangle with side adj. 3″. Find side opp. when the angle is 20°. Tan $= \dfrac{\text{side opp.}}{\text{side adj.}}$

Therfore side opp. = tan x side adj. side opp. = tan 20° x 3″. Side opp. =.36397 x 3″ = 1.0919″. Dia D = 4″ + (2 x 1.0919″). D = 6.1838″.

Problem Diagram (4). A flange for a grinder is mounted on a mandrel between the centres of a lathe. From the given dimensions find at what angle to set the compound rest, to produce the beveled edge. In the right angle triangle, angle A is to be found. Given side adj. = 1½″ and side opp. = ½″.

Tan $= \dfrac{\text{side opp.}}{\text{side adj.}}$ Tan $= \dfrac{.5''}{1.5''} = .3333$. from tables of tangents = 18° 26′ angle A.

Problem Diagram (5). What angle will the compound rest of a lathe be set at to produce an angular recess to the dimensions given? In

the right angled triangle side adj. = 2″. Side opp. = $\frac{6'' - 5''}{2}$ = ½″. Find angle A.

Tan. of A = $\frac{\text{side opp.}}{\text{side adj.}} = \frac{.5''}{2''}$ = .250. From tables of tangents .250 = 14° 2′ angle A.

Problem Diagram (6). A round bar of steel is to be milled to produce a face 1¾″ long. What will be the angle of inclination of the axis of the shaft? Triangle ABC represents the piece removed by milling.

Given. Hyp. AC = 1¾″. Side opp. BC = 1″. Find angle BAC.

Sine = $\frac{\text{side opp.}}{\text{hyp.}} = \frac{1''}{1.750''.}$ = .57143. From table of sines .57143 = 34° 51′ angle BAC.

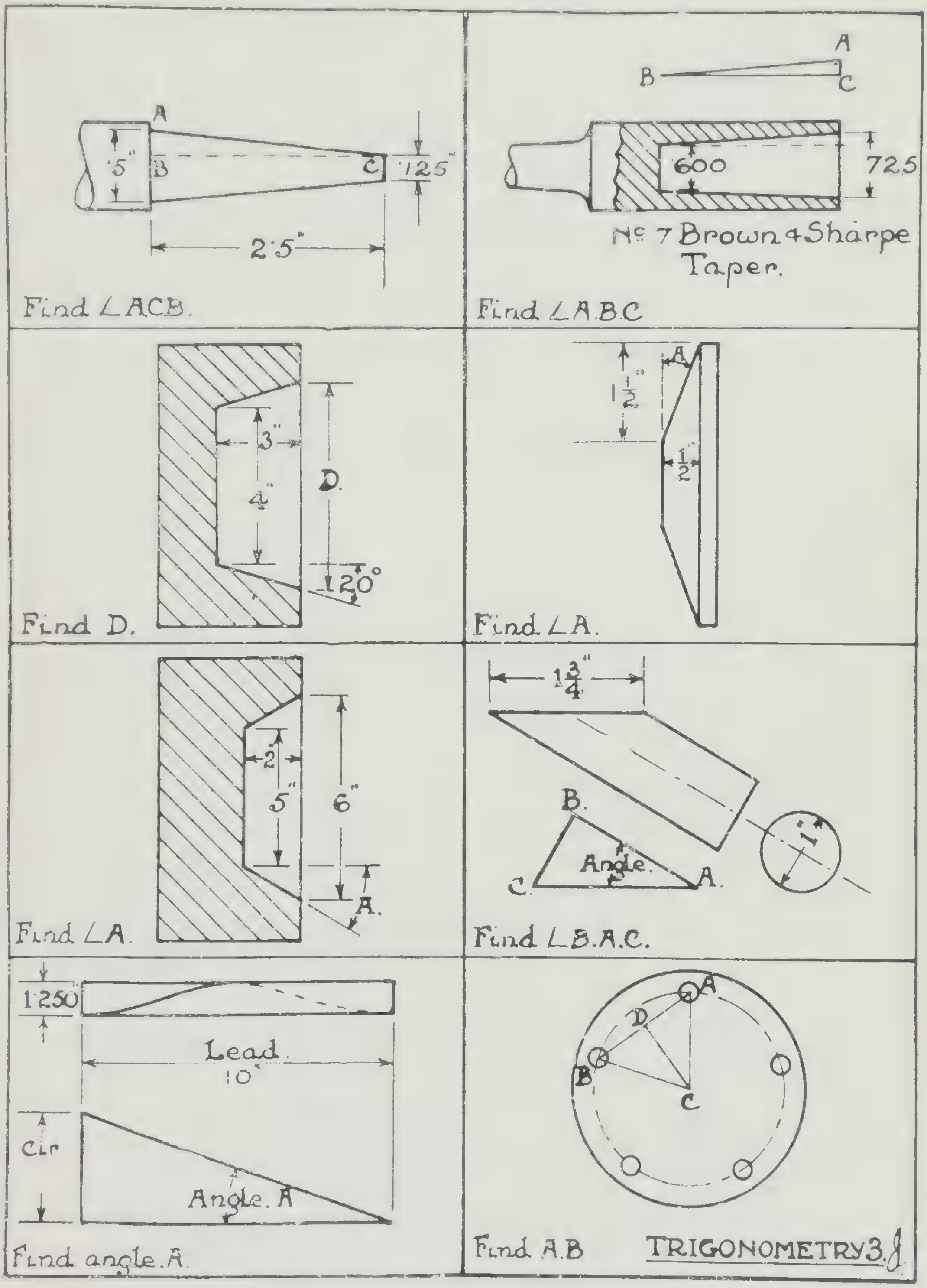
Problem Diagram (7). What angle will the table of a milling machine be set at to mill a spiral to the dimensions given? Given. dia of work, 1,250″, and lead of spiral 10″. Circumference of work = 3.1416 x 1.250″. Circumference = 3.927″ = side opp. Side adj. = 10″. Tan = $\frac{\text{side opp.}}{\text{side adj.}} = \frac{3.927''.}{10''}$ = .3927.

From table of tangents 3927 = 21° 26′. Table is set at angle 21° 26′.

Problem Diagram (8). What dimension will the dividers be set at from a vernier caliper to lay off the chordal measurement on a circle 5″ dia., which is laid out on a flange to locate 5 holes to be equally spaced on the circle? The distance BA is required. First find DA which is one-half BA and is the side opp. in the right-angled triangle ADC. AC is the radius of circle =2½″ and is the hyp. of the triangle.

$$\text{Angle ACD} = \frac{360°}{10} = 36°. \quad \text{Sine} = \frac{\text{side opp.}}{\text{hyp.}}$$

Therefore side opp. = sine x hyp. Sine of 36° = .58779. Side opp. = .58779 x 2.5″ = 1.46947″. BA = twice side opp. DA = 2 x 1.46947″ = 2.93894″.


A
·5"
B
C
·125"
2·5"
Find ∠ACB.
A
B
C
600
725
Nº 7 Brown & Sharpe Taper.
Find ∠A.B.C
3"
4"
D.
120°
Find D.
A
1½"
½"
Find ∠A.
2
5"
6"
A.
Find ∠A.
1¾"
B.
Angle.
C.
A.
1"
Find ∠B.A.C.
1·250
Lead.
10"
Cir
Angle. A
Find angle. A.
A
D
B
C
Find A.B
TRIGONOMETRY 3.

MATHEMATICS

1. If a 70 tooth gear drives a 40 tooth pinion, show that the speed of the driver is inversely proportional to the speed of the follower pinion.
2. In a simple gear drive, 4 gears are used, having 95, 20, 50 and 30 teeth respectively. If the 95 tooth gear is the driver, show by an equation that only the first and last gear affects the ratio of the train.
3. Work out a train of gears for a lathe to cut 10 threads per inch if the lead screw has 4 threads per inch (gear increment 5).
4. What gears will be used on a lathe to cut $11\frac{1}{2}$ threads per inch if the lead screw is 4 pitch? (gear increment 4).
5. Work out a train of gears for a lathe to cut 32 threads per inch with a 4 pitch lead screw (gear increment 5).
6. Work out a train of gears for a lathe to cut 36 threads per inch with a 6 pitch lead screw, and a 2 to 1 dividing gear (gear increment 4).
7. Work out gears for a lathe to cut 8 threads per centimetre with a lead screw 4 threads per inch. A 50,127 translating gear is provided. Sketch the gears as set up. (gear increment 5).
8. If the spindle and stud gear of a lathe has a ratio of 3 to 4, work out a train of gears to cut 8 threads per inch if the lead screw is 6 pitch. (gear increment 5).
9. Work out a train of gears to cut a triple thread $\frac{1}{8}''$ pitch on a lathe with a 4 pitch lead screw. (gear increment 4).
10. Work out a train of gears for cutting a $32''$ lead spiral on a milling machine. The table screw has 4 threads per inch and the worm and worm gear ratio of the index head is 40 to 1. (gear increment 5).
11. Find suitable gears for differential indexing on a milling machine, to index 87 divisions. Selected number as for plain indexing 84.
12. Find a suitable compound gear train to use when differential indexing on a milling machine to index 139 divisions. Selected number as for plain indexing 140.
13. If the return to forward stroke of a shaper is ratio 3 to 2, what will be the cutting speed, if the stroke is $7''$, and the bull gear makes 150 R.P.M.?
14. What R.P.M. must the bull gear of a crank shaper make if the return to forward stroke is 3 to 2 ratio, the stroke is $9''$ and the cutting speed 55 F.P.M. (feet per minute)?
15. What must be the diameter of round stock so that a square bolt head $1\frac{3}{8}''$ across the flats can be milled from it?
16. What is the distance across the flats of a square shank milled the largest possible size from a piece of round stock $1\frac{1}{2}''$ diameter?
17. What will be the bore of a pipe equal in area to two pipes one $8''$ diameter and one $6''$ diameter?

18. How would you find the outside diameter of a gear wheel if only a broken segment of the wheel was available. Illustrate by a sketch.

19. How would you use an index head to index for 26 teeth on a gear to be cut?

20. How would you index for 72 teeth on a gear to be cut?

21. How would you index a plate into 5 degree divisions on a milling machine?

22. How would you obtain linear indexing on a rule to be divided into $\frac{1}{32}''$ divisions accurate to 1/10,000 of an inch, using a milling machine?

23. Three holes are to be located on a plate, the centres of the holes when joined form a right-angled triangle. The smallest angle of the triangle is 23° and the side adjacent to it is $2\frac{1}{2}''$. Find the centre distance between all the holes.

24. Fifteen holes are to be equally spaced on a flange, the centre line diameter of the holes is 5″. Find the distance between the centres of the holes on the circle.

25. A round piece of metal 2″ diameter has 7 flats of equal size milled on it. Find the distance across the corners nearest to being diametrically opposite.

26. What will be the diameter of the large end of a piece of machine steel turned to an inclusive angle of 15° if the length of the taper is 6″ and the diameter of the small end $\frac{3}{4}''$?

27. What will be the distance across the flats of a piece of work when milled from round stock to the largest hexagonal shape, if the round stock is $1\frac{1}{2}''$ diameter?

28. What size round stock would be required to mill flats of the largest hexagonal shape on it if the distance across the flats must be $1\frac{3}{4}''$?

29. What will be the distance across the flats of a piece of stock when milled from round stock to the largest octagonal shape if the round stock is $1\frac{5}{8}''$ diameter?

30. If a triangular template with one angle, a right angle, is to be made with the acute angle 14°-25′, what will be the length of one side adjacent to the right angle if the other side adjacent to the right angle must measure 3″?

31. Find the height of two-sine bar pins above a surface plate when set to test an angle 14°-36′. (Pins are 10″ between centres).

32. If one sine bar pin is 3.198″ higher than the other above a surface plate, what angle is it set at from the horizontal surface plate?

33. If a piece of round stock 4″ long is tapered so that the large end measures 1.127″ and the small end measures .527″, what is the inclusive angle? What angle would the compound rest of a lathe be set at to turn it?

34. What angle would the compound slide rest of a lathe be set at to bore out a socket for a morse taper $5/8''$ per foot?
35. If a tapered recess is bored out in work held in the chuck of a lathe with the compound rest set at 14° and the depth of the recess $7/8''$ and the inside small diameter $2\frac{1}{2}''$, what will be the diameter of the outside of the recess?
36. The outside diameter of a grinder flange being turned in a lathe mandrel is $4''$, and it is to be turned to an angle so that the taper finishes on a boss $1\frac{1}{2}''$ diameter. If the tapered portion is $5/8''$ thick, what angle will the compound slide rest be set at to turn the angle?
37. If a tapered recess is to be made in work held in the chuck of a lathe so that the small diameter of the recess is $2''$, and the large diameter $2\frac{7}{8}''$, what angle will the compound slide rest be set at to bore the work $7/8''$ deep?
38. At what angle would you set a piece of round stock, $3/4''$ diameter, in the chuck of a milling machine Index head, to mill an angular cut which must measure $1\frac{7}{8}''$ across the major axis of the elliptical surface cut across the round stock?
39. What angle would the table of a milling machine be set at to cut a spiral groove in a piece of round stock mounted on centres, if the stock was $1.5''$ diameter, and the lead of the required spiral $17''$? The cutter must be mounted on the arbor.
40. What size would you set a pair of dividers from a vernier caliper with thousandth readings, to set out the distance between 7 holes spaced equidistant on a scribed circle $4\frac{1}{2}''$ diameter on a flange?
41. Calculate the angle at the top and at the root of a triple square thread $1/4''$ pitch to be cut on a piece of stock $1\frac{1}{4}''$ diameter.
42. What will be the lead produced on a piece of work $1\frac{1}{2}''$ diameter which has a spiral cut milled on it, if the angle of the table of milling machine is set correctly for the spiral at 23°? The cutter must be mounted on the arbor.

SHOP SCIENCE

INTRODUCTION TO SPUR GEARING

In this introductory lesson on spur gearing the purpose is to deal very simply with some of the primary features of gearing, to give the beginner sufficient information to allow him to turn up gear blanks in a lathe or form the teeth of a gear on a milling machine.

Size of teeth. A designer may put any number of teeth on a gear. He is limited by the following considerations:—

(1) The ratio of driver and follower gears.

(2) The possible centre distance of the gears.

(3) If strong teeth are required, as in diagram (2), there will be few teeth for a small diameter gear.

(4) Diagram (1) shows smaller teeth, consequently there would be more of them in a gear the same diameter for teeth shown in diagram (2). The greater number of teeth there are in a gear the quieter it will run.

Diametral pitch. The diametral pitch of a gear is the number of teeth there are in a gear for every inch of the pitch diameter. Diagram (1) shows a 10P. (10 Diametral pitch gear) and diagram (2) a 3P (3 Diametral pitch) gear.

Pitch circle and pitch diameter. Diagram (3) shows the two pitch circles of a gear and pinion. The pitch circles represent an imaginary surface which would drive by friction alone. Teeth are cut on gears to prevent slip under load.

Diagram (4) shows a gear with the same pitch diameter as diagram (3) but the "Outside diameter" of the gear is larger than the pitch diameter by an amount equal to 2 addendums.

The module (or measure) is the same part of the pitch diameter as the circular pitch of gear teeth is of the pitch circumference and is equal to the addendum. The module equals the pitch diameter divided by the number of teeth in the gear.

The pitch circle of a gear shown in diagram (3) has 24 teeth divisions, so that the module is 1/24 of the pitch diameter. In a 10P gear, 30 teeth, 3″ pitch diameter, the module is 1/10″ or one divided by diametral pitch.

The outside diameter of a gear equals the pitch diameter + 2 modules.

Gear tooth parts are illustrated in diagram (5).

The circular pitch P′ (P prime) is obtained by dividing the circumference of the pitch circle by the number of teeth in the gear.

The thickness of the tooth (t) is measured on the pitch circle and in cut gears is one half of the circular pitch.

The addendum (S) is equal to the module and is measured on a radial line outside of the pitch circle.

The dedendum is equal to the addendum + the clearance.

The working depth D″ (D second) equals twice the addendum.

The clearance (f) is the space below the working depth and equals one-tenth the thickness of the tooth.

The whole depth of space D″ + f equals the working depth + the clearance.

The fillet or round corner at the bottom of the tooth space is made of equal radius to 1/7 the space between the teeth, measured on the addendum circle.

D = diameter of the addendum circle.

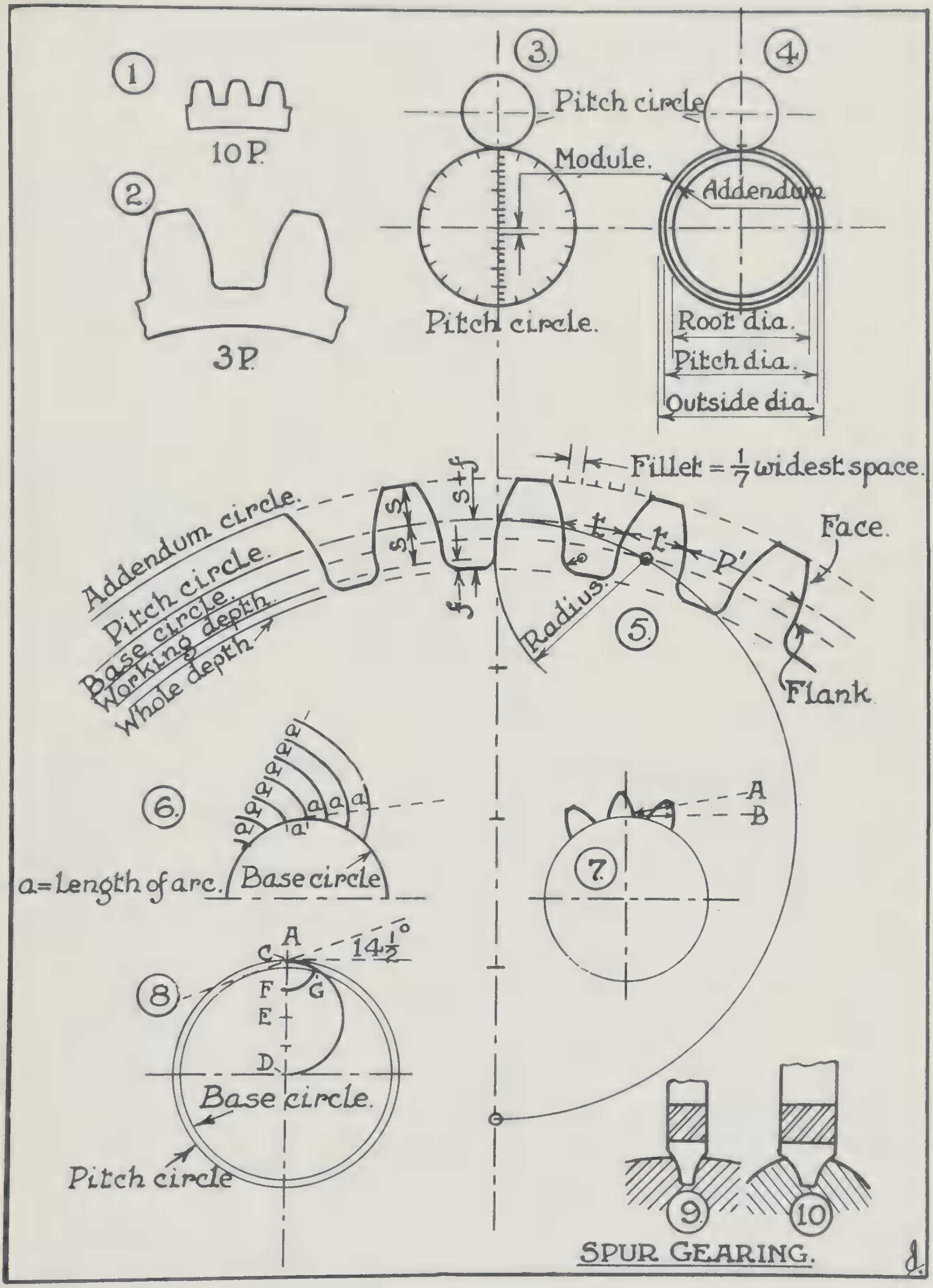
D′ (D prime) = diameter of the pitch circle.

To draw gear teeth (INVOLUTE). Describe a semicircle on the radius of the pitch circle, take ¼ of this diameter and mark it off on the circumference, the point obtained is the centre of the arc to give the tooth form.

Involute form of gear teeth: An involute curve is obtained by unwinding the end of a piece of string from a cylindrical piece of wood equal in diameter to the base circle. The string must be kept taut and the point of a pencil tied on the moving end.

The base circle is a circle of reference and the profile of the tooth outside the base circle is a true involute curve. Diagram (8) shows a method of finding the approximate radius of the base circle for a pressure angle of 14½°. DG. is the radius of the base circle, when CG equals ¼ of pitch circle radius struck on semicircle C.G.D. the diameter of which is ½ diameter of pitch circle.

Diagram 6 shows several involute curves. It will be observed that distance (a) always measured at a tangent to the base circle is constant wherever measured. The profile of gear teeth can be tested by use of an Odontometer which gives a practical application of the distance (a) in diagram (6). This is further illustrated in diagram (7) when distance A and B should be equal, as checked by an Odontometer. Shape of teeth varies with the number of teeth in a gear for same diametral pitch. Diagram 9 shows a No. 1 cutter to cut 135T to rack, and diagram 10 shows No. 8 cutter to cut 12 to 13T. A set of eight cutters for each pitch will cut all sizes of gears from 12T to a rack, but half sizes may be obtained by special cutters.


1
10 P.
2
3 P.
3
Pitch circle
Module.
Pitch circle.
4
Addendum
Root dia.
Pitch dia.
Outside dia.
Fillet = 1/7 widest space.
s+f
s
s
f
t
t
P'
Face.
Addendum circle.
Pitch circle.
Base circle.
Working depth.
Whole depth.
Radius.
5
Flank.
6
a
a'
a = length of arc.
Base circle
A
B
7
8
A
C
14½°
F
G
E
D
Base circle.
Pitch circle
9
10
SPUR GEARING.

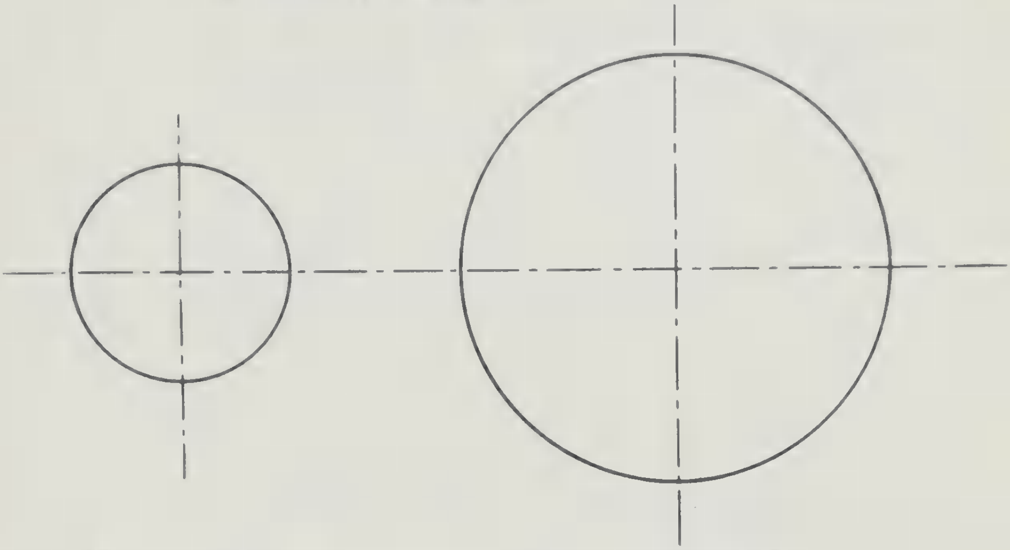
GEAR TOOTH PROPORTIONS

(Diametral Pitch)

Diametral Pitch is the number of teeth to each inch of the pitch diameter.

Find	*Rule*
The Diametral Pitch.	Divide 3.1416 by the Circular Pitch.
The Diametral Pitch.	Divide number of teeth by Pitch Diameter.
The Diametral Pitch.	Divide number of teeth plus 2 by Outside Diameter.
Pitch Diameter.	Divide number of teeth by the Diametral Pitch.
Pitch Diameter.	Subtract from the Outside Diameter the quotient of 2 divided by the Diametral Pitch.
Pitch Diameter.	Multiply the Addendum by the number of teeth.
Outside Diameter.	Divide number of teeth plus 2 by the Diametral Pitch.
Outside Diameter.	Add to the Pitch Diameter the quotient of 2 divided by the Diametral Pitch.
Outside Diameter.	Divide the number of teeth plus 2 by the quotient of number of teeth and by the Pitch Diameter.
Outside Diameter.	Multiply the number of teeth plus 2 by the Addendum.
Number of Teeth.	Multiply Pitch Diameter by the Diametral Pitch.
Number of Teeth.	Multiply Outside Diameter by the Diametral Pitch and subtract 2.
Thickness of Tooth.	Divide $\frac{3.1416}{2}$ or 1.5708 by the Diametral Pitch.
Addendum.	Divide 1 by the Diametral Pitch.
Root.	Add 1-tenth the thickness of tooth to the Addendum or Divide 1.157 by Diametral Pitch.
Working Depth.	Divide 2 by the Diametral Pitch.
Whole Depth.	Add 1-tenth the thickness of tooth to working depth or Divide 2.157 by Diametral Pitch.
Clearance.	Divide .157 by the Diametral Pitch or take 1-tenth of thickness of tooth.

PROBLEM. **Find** proportions for a gear.
Given the mating pinion which has half the number of teeth of the gear.



Given Pinion 50 Teeth
Outside diameter—4″
(Obtained by measurement)
Number of teeth—50
(Obtained by counting)

Gear to be made
(Work out proportions)

Name of Part:				
Number of teeth in gear	N			100
Outside diameter of gear	D	$\frac{N+2}{P}$	$\frac{100+2}{13}$	7.8461″
Diametral pitch (from) (pinion)	P	$\frac{N+2}{D}$	$\frac{50+2}{4}$	13
Circular pitch	P′	$\frac{3.1416}{P}$	$\frac{3.1416}{13}$	.2417″
Thickness of tooth	t	$\frac{1.5708}{P}$	$\frac{1.5708}{13}$	.1208″
Addendum and module	s	$\frac{1}{P}$ or $\frac{D'}{N}$	$\frac{1}{13}$	.0769″
Working depth of tooth	D″	$\frac{2}{P}$	$\frac{2}{13}$	.1538″
Depth of space below pitch line		s+f	.0769″ +.0120″	.0889″
Clearance	f	$\frac{t}{10}$	$\frac{.1208''}{10}$	.0120″
Whole depth of tooth		D″+f	.1538 +.0120	.1658″
Pitch diameter	D′	$\frac{N}{P}$	$\frac{100}{13}$	7.6923″
Centre distance		$\frac{D' \text{ of gear}}{2}$ +	$\frac{D' \text{ of pinion}}{2}$	5.769″

SPUR GEARS (INVOLUTE)

THE VERNIER

A vernier used in conjunction with a graduated scale will help to produce measurements of a much finer degree than is possible by means of a graduated scale alone. The degree of accuracy will depend upon the graduations on the main scale and the graduations on the vernier.

Diagram (1). Illustrates a simple vernier, which will serve to explain the functions of a vernier for one who does not understand its use. If four vertical cuts are made with a knife at A and B in diagram (1), and the strip in diagram (2) is cut out to slide between these strips A and B, trial readings may be made to test ones ability to read a simple vernier.

The vernier principle. The length of the vernier scale is made equal to a definite part of the main scale, but the divisions on the vernier scale are usually graduated to 1 space less than the spaces on the main scale, but the same number of divisions are used.

Example Diagram (1). The main scale 1″ has 10 divisions. The vernier scale from 0 to 10 is equal to 9 of the divisions on the main scale, that is, one less than 10 and the vernier scale is divided into 10 equal parts.

To read the vernier. If 0 and 10 correspond with marks on the main scale, the measurement will be in tenths only. If the first mark (the one next to 0) on the vernier scale corresponds to a mark on the main scale, no other mark will correspond and the vernier is advanced 1/10 of 1/10″ = 1/100″ = .01″. Similarly if the 6th mark corresponds the vernier is advanced 6/10 of 1/10″ = 6/100″ = .06″.

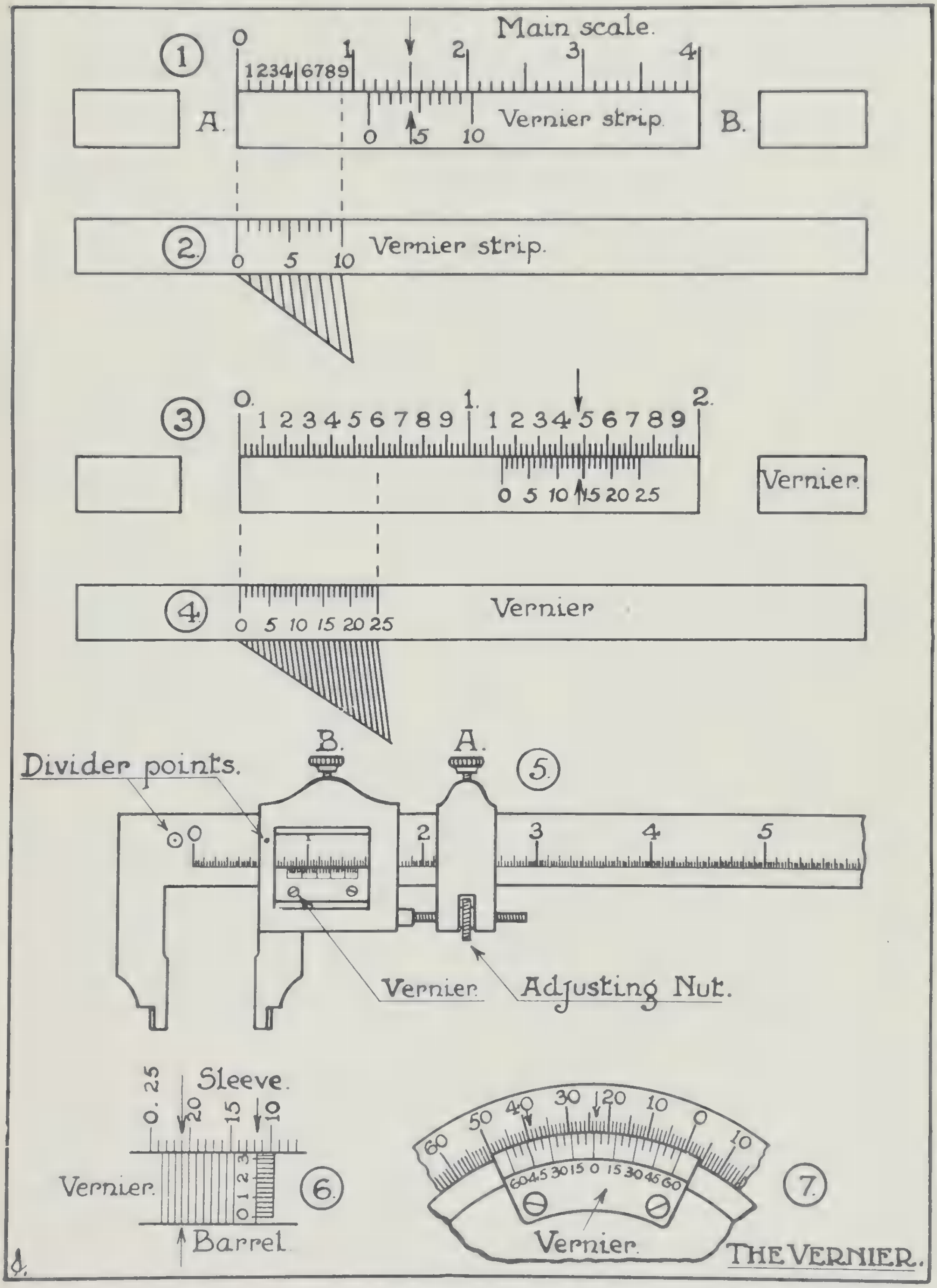
In Diagram (1) the reading is as follows. The zero mark on the vernier scale is past 1″ and 1/10″ and the fourth mark on the vernier scale corresponds.

Reading	main scale	1.1″.
	vernier	.04″.
	Total	1.14″.

A vernier to read in thousandths of an inch. Diagram (3). This is drawn twice full size to assist in reading the small divisions. Cut out strip (4) and slide in cut strips in (3) as previously described.

The vernier scale 0 to 25 is equal to 24 small divisions on the main scale and is divided into 25 parts, because it is made 1 less than 25 parts on the main scale.

The main scale is divided into tenths, then subdivided into fortieths. 1/40″ = .025″ or 25 thousandths. If the first mark on the vernier scale


Main scale.
A.
B.
Vernier strip.
Vernier strip.
Vernier
Vernier
Divider points.
B.
A.
Vernier
Adjusting Nut.
Sleeve.
Vernier.
Barrel
Vernier.
THE VERNIER.

concides with any mark on the main scale the vernier is advanced 1/25 of .025″ = .001″ or one thousandth of an inch.

Reading the thousandth vernier. Diagram (3). The zero mark on the vernier scale is past 1″ and 1/10″ and 1/40″ and the 14th mark on the vernier corresponds with a mark on the main scale.

Reading is therefore:—

Main scale	1.0″
Main scale	0.1″
Main scale	.025″
Vernier	0.014″
Total	1.139″

Diagram (5) shows a drawing of a common vernier caliper for one thousandth reading. *To adjust.* The screw A is tightened and the screw B loosened; then a fine movement of the sliding jaw may be obtrained by turning the vernier adjusting nut. A magnifying glass will be found a great assistance in reading accurately the fine division on the vernier. A distance may be set on dividers for accurately laying out work by placing one leg of the dividers in the spot inside the circle shown on the fixed jaw, and the other in the conic recess in the sliding jaw. The jaw points are designed for measuring internal dimensions with their equivalent zeros on the reverse side of the scale.

Vernier on a micrometer. Diagram (6). This provision allows one to read a micrometer accurately to 1/10,000″. *Reading* tenths on barrel 0.300″. Fortieth on barrel 0.025″. Thousandths on sleeve 0.011″. Graduations on vernier 0.0007″. Total 0.3367″.

Vernier on a bevel protractor. Diagram (7). From 0 to 60 on the vernier is divided into 12 divisions, which are equal to 23 divisions on the main scale. One division on the vernier is 1/12 of a degree shorter than two divisions on the main scale 1/12 of 1 degree (or 60 minutes) equals 5 minutes. Reading diagram (7). Main scale = 23°. Vernier = 8 x 5 = 40′. (The 8th mark corresponds). Total = 23° 40′.

HEAT TREATMENT DEFINITIONS

The following definitions have been approved by the Recommended Practice Committee of the American Society for Steel Treating:

(1) Annealing. Heating above the "critical temperature" followed by a relatively slow rate of cooling.

By "critical temperature" is meant that temperature which is customarily associated with the following phenomena:

(a) Hardening when quenched. (b) Loss of magnetism. (c) Absorbtion of heat. (d) Formation of solid solution. (e) Pronounced refinement of coarse grain upon cooling.

(2) Loneal (A new term). Heating below the "critical temperature" followed by any rate of cooling.

(3) Normalizing. Heating above the "critical temperature" followed by an intermediate rate of cooling.

(Note: In good practice, the heating is considerably above the "critical temperatures").

(4) Spheroidizing. A long time heating at or about the "critical temperature" followed by slow cooling throughout the upper part of the cooling range.

(Note: For the purpose of spheroidizing the cementite in high carbon steels.)

(5) Hardening. Heating above the "critical temperature" followed by a relatively rapid rate of cooling.

(6) Tempering. Reheating after hardening to some temperature below the "critical temperature" followed by any rate of cooling.

(7) Carburizing. Adding carbon with or without other hardening elements, such as nitrogen, to wrought iron or steel by heating the metal below its melting point in contact with carbonaceous material.

(8) Case hardening. Carburizing the surface portion of an object and subsequently hardening by suitable heat treating.

(9) Cyaniding. A specific application of carburizing where the object, or a portion of it is heated and brought into contact with cyanide salt.

Carbon steels suitable for various uses. Carbon content 0.65 to 0.85 per cent. Shear blades, boiler snaps and cups, hammers, stamping and pressing dies, mining drills.

Carbon Content 0.81 to 0.95 per cent. Hot and cold sets, chisels, dies, shear blades, mining drills, smith's tools, set hammers, swages, flatteners.

Carbon Content 0.96 to 1.10 per cent. Small cold chisels, hot sets, small shear blades, large pinchers, large taps, granite drills, trimming

dies, turning tools, planer tools, drills, cutters, slotting and milling tools, mill picks, circular cutters, small shear blades, threading dies.

Carbon Content 1.11 to 1.25 per cent. Small cutters, small taps, drills, slotting and planing tools, wood-cutting tools, turning-tools, razors, etc.

Suitable tempering heats for various tools. Temperature 350° to 390° Fahr. (177° to 199° Cent.). Lathe tools for brass and copper alloys. Milling cutters for brass and copper alloys. Scraper and cutting tools for soft metals. Drawing mandrels, drawing dies, bone-cutting tools, hammer faces, steel engraving tools, wood-carving tools, cutting tools for iron and steel, hand tools, threading dies for brass.

Temperature 400° to 500° Fahr. (204° to 260° Cent.). Hand taps and dies, hand reamers, drills, bits, cutting dies, penknives, milling cutters, chasers, inserted saw teeth, press dies sheet steel, rock drills, taps and dies, wire drawing dies, dental and surgical instruments, twist drills.

Temperature 500° to 600° Fahr. (260° to 315° Cent.). Bending and forming dies, shear blades, chuck jaws, forging dies, drifts, ,gauges, press dies, flat drills, reamers, chisels and tools for wood cutting, hammers and drop dies, axes, lathe tools for copper, augers, cold chisels, coppersmith tools, grinders, screw drivers, molding and planing tools, hacksaws, needles, butcher knives, saws.

TABLES OF HEAT TREATMENT

Normalizing of Tool Steel Before Hardening

Operations:

(1) Heating. Place the steel in a furnace so as to expose maximum surface area. Heat uniformly to a temperature above the upper critical range as shown in Table I and hold at this temperature for sufficient time to obtain complete penetration of heat and for complete refinement of grain.

(2) Cooling. Remove from furnace and cool freely in air.

Table I

Normalizing Temperature and Carbon Range.

Carbon	Degrees Fahr.	Degrees Cent.
0.65 to 0.90 per cent.	1475 to 1525	801 to 829
0.80 to 0.95 " "	1475 to 1500	801 to 815
0.95 to 1.10 " "	1500 to 1575	815 to 857
1.10 to 1.25 " "	1575 to 1650	857 to 898

Cross Sections, Weight and Time.

Thickness of Largest Section of Unit	Weight of Unit in Pounds (approximate)	Time of Heating in hours (approximate)	Time of Holding in hours (approximate)
Up to and including 1 inch	Up to 100	¾	½
Over 1 inch and including 2 inches	Over 100 and including 300	1¼	½
Over 2 inches and including 3 inches	Over 300 and including 500	1¾	¾
Over 3 inches and including 4 inches	Over 500 and including 1,000	2¼	1
Over 4 inches and including 5 inches	Over 1,000 and including 5,000	2¾	1
Over 5 inches and including 8 inches	Over 1,500 and including 2,000	3½	1½

Heat Treating of Plain Carbon Tool Steel.

(1) Heating for quenching. Heat steel uniformly to the temperature indicated in Table II.

Table II

Heat Treating.

Carbon Range Per Cent.	0.65—0.80	0.81—0.95	0.96—1.10	1.11—1.25
Hardening Temperatures	1550—1450 Deg. Fahr.	1460—1410 Deg. Fahr.	1390—1430 Deg. Fahr.	1380—1420 Deg. Fahr.
Quenching Medium and its temperature	Water at 70 Deg. Fahr.	Water at 70 Deg. Fahr.	Water at 70 Deg. Fahr.	Water at 70 Deg. Fahr.

(2) Quenching. Quench from the temperature in Table II in water but do not cool below the temperature of boiling water 212° Fahr. (100° Cent.).

(3) Tempering. Reheat immediately in oil, liquid heating medium or furnace for the time and at the temperature specified in Table III and cool.

Table III

Tempering the Tool Steel.

Results Desired	Tempering Medium	Temperature
Relieving Strains	Oil	350 to 375 Deg. Fahr.
Relieving Strains and Reducing Brittleness	Oil or salt	400 to 500 Deg. Fahr.
To Relieve Strains and to Toughen	Oil or salt	500 to 600 Deg. Fahr.

HEAT TREATMENT OF CARBON TOOL STEEL

(Line Diagrams)

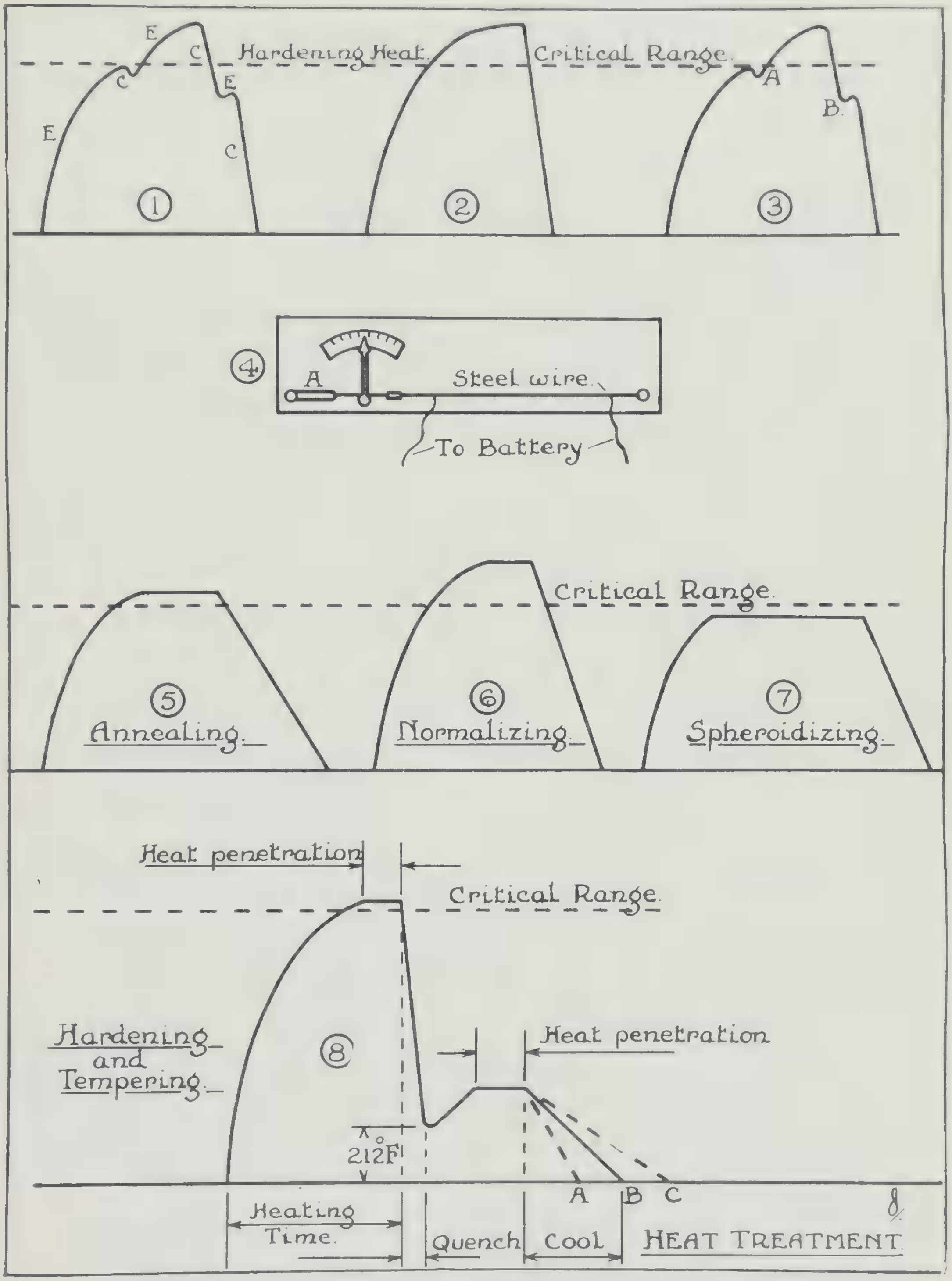
This lesson presents a simplified outline of the common forms of heat treatment, any of which have been defined as "an operation or combination of operations involving the heating and cooling of metal in the solid state." The illustrations are in the form of line diagrams or graphs, which picture the operation graphically and if these pictures are carried in one's mind, a definite plan of action should be established. Perhaps the most important single fact that should stand out in any operator's mind is that of the "Transformation points" or "Critical periods".

Decalescent and Recalescent points. Experiment Diagram (4). The phenomena of Decalescent and Recalescent periods can best be shown by diagram (1) which fact is shown clearly by a simple experiment which is illustrated in diagram (4). A piece of steel wire is fastened to a terminal at one end and wrapped around the drum of an indicator and stretched by means of a piece of rubber at A. Two wires connect the steel wire to a battery which is used to heat the wire. On heating, the wire expands, then suddenly contracts although the wire is getting hotter steadily. This action is clearly shown by the indicator. When the heat is cut off the wire cools and contracts but suddenly the indicator shows it expand, then it contracts again. Diagram (1) illustrates this action, E representing the expansion and C the contraction. The high critical point is known as the Decalescent period and the low change point as the Recalescent period.

The solution of this peculiar action lies in the fact that the heat or energy instead of increasing the temperature of the steel is used up in making an internal structural and chemical change. The carbon in the metal goes into a solid solution with the ferrite or iron and produces the austenite condition.

Diagram (2) shows the graph of the heating and cooling of a furnace. If a thermocouple were placed in a drilled hole inside a piece of tool steel and heated in the same furnace, the pyrometer attached to the thermocouple would show the critical periods (diagrams 3) of Decalescent at A and Recalescent at B. The importance of this is due to the fact that the proper hardening heat of the steel is just after the Decalescent period or as soon as the heat starts to rise again. If the steel is quenched from this heat into water, file hardness is obtained and the steel will have the finest grain structure. If quenched from a temperature much above the hardening heat, the grain of the steel will be coarse, crystalline and weak. If allowed to cool off and quenched on a falling heat, the grain will still be the same as if the steel were quenched from the higher temperature. If the steel is allowed to cool off below the Recalescent point (B), no hardness will be obtained when quenched.

Annealing. Diagram (5). The line diagram illustrates the idea that the annealing process consists of heating the steel to a temperature greater than the hardening heat and holding that temperature for some time. then cooling very slowly. Annealing is a very broad term and can be carried out in many ways to obtain varying results. The results will


E
E
C
C
E
C
Hardening Heat.
Critical Range.
A
B.
1
2
3
4
A
Steel wire.
To Battery
Critical Range.
5
Annealing.
6
Normalizing.
7
Spheroidizing.
Heat penetration
Critical Range.
Hardening
and
Tempering.
8
Heat penetration
212°F
A
B
C
Heating
Time.
Quench
Cool
HEAT TREATMENT.

depend upon the rate of heating, the maximum temperature and the rate of cooling.

The purpose of annealing is to remove stresses, to induce softness and to refine the crystalline structure. The cooling rate can be controlled, by cooling off with the heated furnace, or burying the steel in lime, or, if a forge is used, covering the steel with coals and allowing it to cool with the fire. When slowly cooled from above the critical range, the solid solution (austenite) in the steel will break up into a mixture of crystalline grains of free ferrite and pearlite. If the steel contains about 0.85% carbon, the structure will be roughly 100% pearlite after annealing. If the steel contains over 0.85% carbon, the carbide of iron (cementite) occurs as a free constituent with pearlite.

Normalizing. Diagram (6). The normalizing treatment is for the purpose of removing strains put in the steel by forging and should always precede hardening and tempering, to guarantee that all strains have been removed, which will lessen the possibility of cracks occurring in the hardening process. The steel is heated to a temperature much higher than the hardening heat or the annealing heat and held there for a short time only. The cooling rate is comparatively rapid when compared to annealing. After heating, the steel can be cooled off in the air. The microstructures obtained consist of Sorbite with or without free ferrite or cementite, according to the carbon content. Steels below 0.85% carbon after slow cooling through the critical range consist of a mixture of pearlite and free ferrite.

Spheroidizing. Diagram (7). This process is usually applied to steel of high carbon content to make it easy to machine. The steel is usually heated for a considerable time at a temperature just below the critical range, followed by slow cooling. The object of this heat treatment is to produce in the steel a globular condition of the carbide which is hard to machine if in layers, but if in globular form, the hard globules are pushed by the tool into the soft ferrite as the tool enters and consequently it machines easily.

Hardening and tempering. Diagram (8). The line diagram illustrates the fact that the steel is heated uniformly and held at the hardening heat just beyond the critical range, the time for heat penetration depending upon the mass of metal being heated. The steel is then quenched rapidly, the speed with which the heat is removed will decide the hardened condition of the steel. Steel may be produced in the Martensite, Troostite or Sorbite condition, according to the rate of the quench. It is quite unnecessary to quench steel outright to room temperature as the lower temperature zone is the cracking zone. If the steel is quenched to the temperature of boiling water—212° F, there will be less likelihood of cracking taking place and still it will be file hard if quenched in water or proper quenching oil.

It is best to reheat immediately to "temper" or "draw" which will reduce the hardness to a proper degree for the work the tool has to do. Strains will be relieved and the metal made hard and tough, according to the tempering temperature. The steel should be held at the tempering temperature for some time to guarantee penetration of the heat and then cooled slow or fast, according to the condition of the steel required.

HARDENING STEEL

Any person who lacks experience in hardening would do well to conform to certain simple rules to prevent him from certain failure in his work. There are so many things that may happen that unless due consideration is given to certain natural laws, trouble is bound to be met with sooner or later.

Decarbonized steel. It is quite common to see some people attempting to harden an odd tool in a forge fire with very little fuel and the blast wide open. Steel heated in such a fire will receive heat on the bottom surface at a tremendous rate while the inside of the metal is practically cool. Diagram (1) shows that the effect of this sort of treatment is to burn out the carbon on the surface of the metal exposed to such heat and, when the body of the metal has been heated through and quenched, the decarbonized surface cannot possibly be hardened, because the carbon which is the main hardening element has been burnt out of it.

Detection of decarbonized steel. Apply steel to a grindstone. A properly hardened steel shows the carbon spark, as in Diagram (2), while a decarbonized steel shows an iron spark, as in Diagram (3).

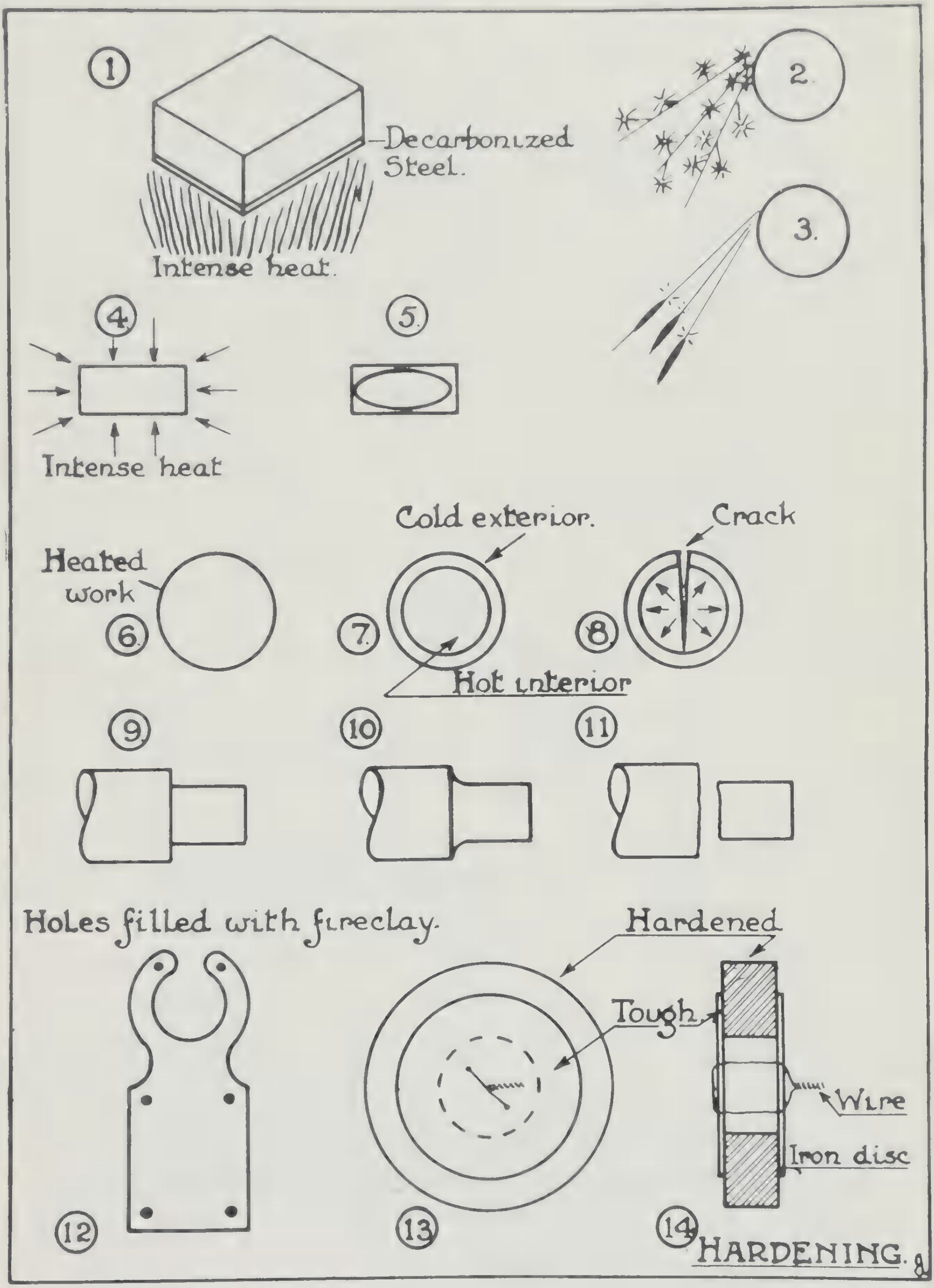
Internal strains in steels due to careless heating. Hardening cracks are often caused by the steel being heated too quickly, that is, the outside of the steel is subject to an intense heat while the inside lags, as shown in Diagram (4). The outside expands very rapidly and the inside remains more or less in a contracted state; consequently separation due to the intense internal strain takes place, as shown in Diagram (5). This sample was taken from a broken cape chisel showing a perfect crack in the shape of an ellipse.

The effect of careless quenching. Diagrams (6), (7) and (8) show views in section of centre punches during the hardening process while using a gas forge or coke forge. In Diagram (6) the steel is heated evenly throughout to the hardening heat, but is quenched right out in cold water. The effect of this, as shown in Diagram (7), is to freeze the outside of the metal quickly, sealing the metal inside which takes some time to cool. The inside, when cooling, expands on account of the formation of hardening crystals, in the same manner that water expands when freezing, and consequently bursts open the steel, as shown in Diagram (8). To prevent this the steel should never be quenched beyond the temperature of boiling water .212 F.; or better still, remove from the water as soon as it shows a black heat, and finish cooling in oil, to give a slow rate of cooling and freedom for the internal strains to work out; then temper immediately.

Shaping work to prevent "hardening cracks". If tool steel were formed as shown in Diagram (9) there would be a distinct line of separation between the greater and the smaller mass of metal. During cooling in the hardening process the smaller part will cool before the larger part, and the strains will tend to separate the metal at the sharp corner, as shown in Diagram (11). To offset this tendency all corners, where possible, should be filleted, as shown in Diagram (10), so that the strain between the smaller and the larger diameter will be more gradual. If a sharp corner is necessary, it can be ground after hardening.

Filling of holes to prevent "hardening cracks". A hole through a piece of metal allows the coolant to quench the steel quickly long before the remaining mass of metal, as shown in the punch in Diagram (12). If these holes shown are filled with fireclay before heating, the metal will cool at the same rate approximately as if no holes were there, and thus reduce cracking tendencies.

To harden one part and toughen the remainder. This can be accomplished, as shown in Diagrams (13) and (14), by protecting the part to be toughened with two metal plates wired together before hardening. When quenching, the plates prevent the cold water from gaining access quickly to the surfaces requiring protection and thus form a blanket of steam or warm water, which delays quenching, so that while the outside quenches quickly and is file hard, the inside quenches slowly and is consequently toughened.


1
Decarbonized Steel.
Intense heat.
2.
3.
4
5.
Intense heat
Cold exterior.
Crack
Heated work
6.
7.
8.
Hot interior
9
10
11
Holes filled with fireclay.
Hardened
Tough
Wire
Iron disc
12
13
14
HARDENING.

HEAT TREATMENT WITH MODERN EQUIPMENT

Steel hardened in an ordinary forge fire or gas forge, is exposed to many things that tend to produce poor results. (1) Decarbonization of the outside of the work due to direct contact with the fire or flame. (2) Irregular rate of heating. The outside of the steel may be beyond the hardening heat before the inside of the steel has reached it. The effect of this kind of treatment is to produce work below the required maximum hardness, warping and hardening cracks, unless great care is exercised. An even rate of heating throughout the piece and exact hardening heat temperature are of supreme importance if satisfactory results are to be obtained. *The Hardening furnace* provides suitable conditions for heating work for hardening. The electric furnace produces the best type of heat, while gas and oil-fired furnaces are commonly used.

The furnace illustrated in Diagram (1) is suitable for gas or oil. It consists principally of two chambers. The lower is the combustion chamber and the upper chamber or muffle is where the work is placed, being protected from the products of combustion. The door of the furnace is moved vertically up and down and protects the work from oxidation from the outside air. A peep hole and cover is provided so that the operator can observe the heat of the work.

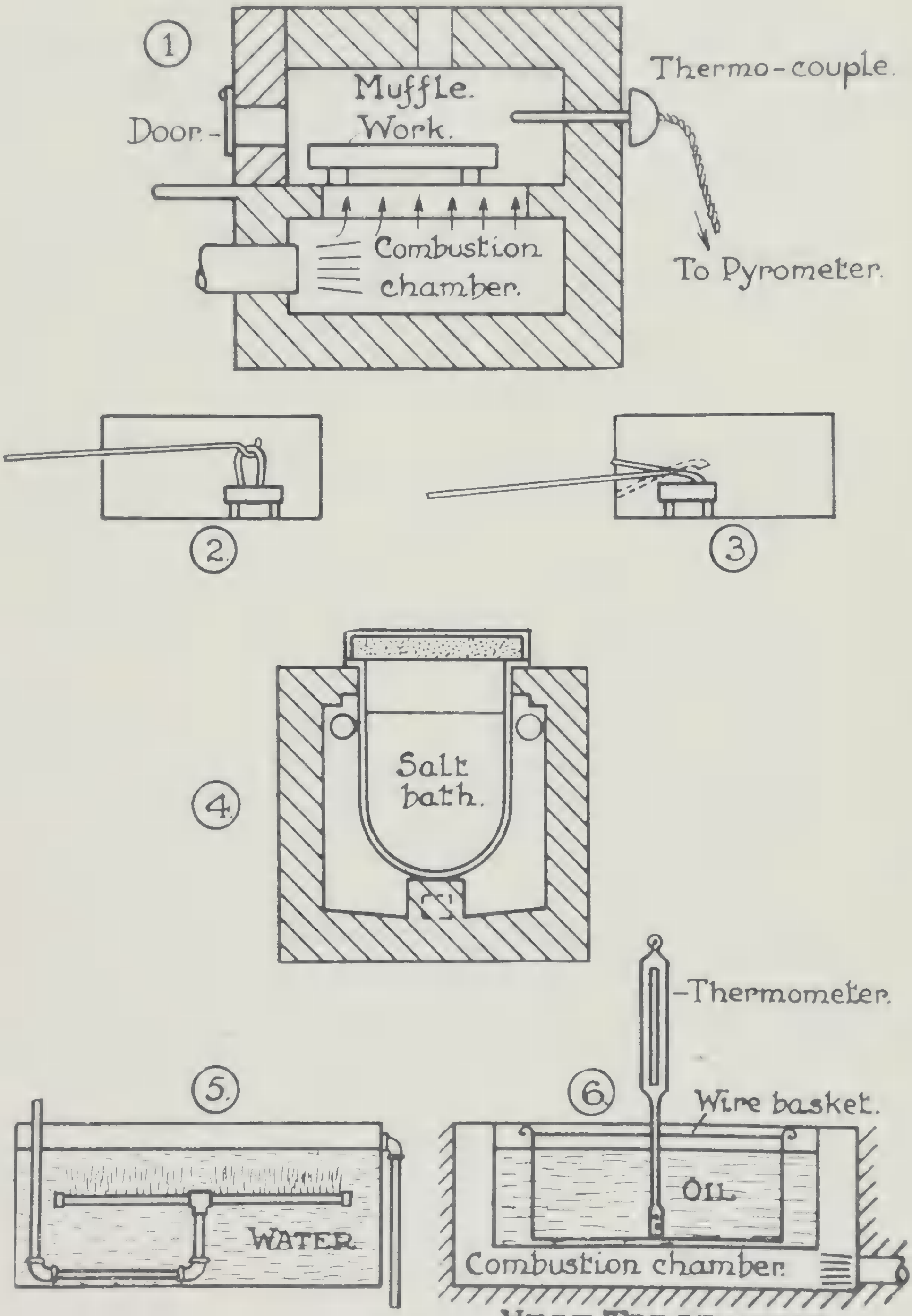
A thermocouple is shown at the back of the muffle. The hot end or junction of two wires, one platinum and one platinum rhodium, is protected by a protective tube of a material called "stonax", which has a very high melting point.

The heat passes up from the combustion chamber at the two sides and through the vent at the top of the furnace. By watching the appearance of the gases at the vent the operator can tell if the mixture of gas and air in the combustion chamber is correct.

The Pyrometer is usually placed at a convenient point for reading the temperature; but one should bear in mind that the thermocouple is very sensitive and registers heat quickly, while the work usually lags behind, so that allowance must be made for the time required for the work to absorb its heat for the temperature required.

The work should be placed in the furnace so that all parts have an equal chance to heat regularly. Sometimes it is advisable to rest the work on raising blocks to produce even heating.

Testing for hardening heat. When the steel reaches its "critical point" or "decalescent point" it becomes non-magnetic. The hardening heat is correct just after it passes this point, so that a magnet can be



HEAT TREATMENT.

placed inside the furnace, as shown in Diagram (2), to feel if the steel is magnetic. If the magnet does not "pull" close the door for a while to regain the lost heat, then remove the work and quench. A lever magnet as shown in Diagram (3) could be used for the same purpose.

Liquid baths for heating. Diagram (4) shows a common type of liquid bath used for heating steel. The advantage of such heat treatment over an ordinary furnace is that it provides a non-scaling and non-oxidizing method of heating the work. The bath heats the work uniformly and eliminates warping and cracking. The liquid medium used may be one of two classes: (1) *metals,* such as lead (which is commonly used (2) *salts,* which are very numerous and give satisfaction.

Quenching. A suitable quenching bath should be provided, similar to the one shown in Diagram (5). The temperature of the bath for certain kinds of work is important; but generally a temperature of 45° F. is suitable. By arranging the water intake and overflow pipes, as shown, the bath is kept at a constant temperature and the water is kept moving to give an even quench on the work. Oil is sometimes used to quench certain tools, which would likely crack owing to their shape if quenched in water. The oil gives a slower rate of cooling than the water; therefore the work is not so hard as when water-quenched, but it is less brittle.

Note. When quenching, the work should never be quenched right out or cold, but should be removed from the bath while at a black heat and tempered immediately. (Feel the work under water).

The tempering bath, Diagram (6), usually consists of special tempering oil or salt in a tank with provisions made for heating by a surrounding combustion chamber. A thermometer is placed in the heated oil or salt to notify the operator of the temperature of the bath. The work is submerged in the oil in a wire basket until the proper temperature has been reached and then cooled off at any rate of cooling until cold.

OUTLINE OF HEAT TREATMENT

Steel often has to be heated to be forged to shape for tools. The steel can be improved or damaged by careless treatment. The finest grain size in steel can be obtained just at, or just above the critical range, there fore all hammering should stop when the steel is at this temperature.

Hot working. A steel ingot shows a coarse grain, but by hammering at the proper heat the grain size may be reduced. The pressure of rolling or hammering must be sufficient to penetrate to the centre of the steel to obtain a fine grain size throughout.

The steel should be annealed or normalized after working to remove the strains and to give it the proper condition, so that when hardened, a fine silky grain is produced.

Cold working of steel below the "critical range" tends to distort the steel and reduce its ductility and causes some hardness and brittleness. The advantage of cold working is that no oxide is formed on the metal and the comparatively hard surface produced usually makes it suitable to withstand friction and wear.

Softening treatment for steel. Steels may be softened to remove strains put in by working, or to remove hardness from some previous hardening process, or to give it a fine grain preparatory to hardening.

There are two general processes for softening steels. (1) Annealing (2) Normalizing.

Annealing is a heat treatment usually given by packing work in sealed boxes, which produces a maximum amount of ductility combined with a high elastic limit. All strains are removed and the steel is given good machining qualities. The work is left to cool off in the furnace to give a very slow rate of cooling, or the work can be removed from the furnace and buried in lime or ashes.

Normalizing is a heat treatment beyond its highest critical temperature and soaking at this heat to relieve crystal formation under strain from forging. The temperature for normalizing is usually much higher than for annealing. When heating for a long period, as in annealing, there is a great tendency for the outer part to oxidise. This scale (iron oxide) so formed by heating means that the outer part of steel has decarburized. For this reason the work is placed in sealed boxes to keep out the air and the pots are filled with the work placed between layers of a material rich in carbon.

Hardening treatment. When a cold piece of steel is to be hardened and is placed in a furnace for that purpose, the pearlite and ferrite or cementite will be changed to austenite.

The ferrite above the critical point changes and dissolves the cementite and forms a "solid solution" called "Austenite." If this process is rushed by placing cold work in a hot furnace, a molecular expansion occurs which the steel cannot withstand and it cracks. Similar results are obtained by impinging a flame on steel. Work should therefore be pre-heated before putting into a hot furnace, or salt baths should be used.

Quenching. If work is quenched from just past its high critical point, the Austenite tends to revert rapidly through its various stages back to pearlite, but it is trapped during the process by the rate of the cooling, and if the cooling is rapid the resulting structure is composed of Martensite, which is very hard, and in the condition of greatest volume and finest grain structure.

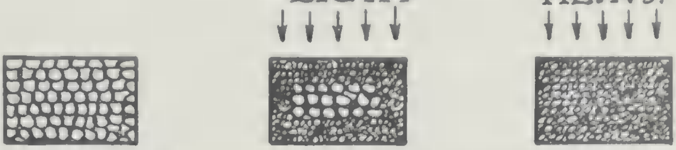
If the quench is less rapid, such as is obtained by quenching in oil, Troostite is formed with the result that the steel is not so hard as the Martensite condition. Sometimes a large piece of steel may have Martensite on the outside and Troostite in the centre on account of the reduced rate of the cooling inside the large piece of steel. If work is quenched with a very slow rate of cooling, Sorbite is formed with the steel in a strong and ductile condition.

Strengthening treatment. After hardening the steel may be reheated to remove its brittleness which accompanies the Martensite condition. This reheating or Tempering allows the constituents of the steel to go on with the change from Martensite to Troostite and so on to Sorbite and finally pearlite. By regulation of the temperature the steel may be left in any condition as required for the work in hand.

Strains may therefore be relieved, brittleness taken out and the steel left with the proper amount of hardness and toughness desired.

LIGHT HEAVY

HOT WORKING.



STEEL INGOT. ROLLED or HAMMERED.

SOFTENING TREATMENT — PEARLITE.

1. ANNEALING.
To produce soft metal with fine grain size.

2. NORMALIZING.
To relieve strains from forging and mechanical work.

HARDENING TREATMENT.

1. Ferrite and Cementite, below critical point.
Above the critical point.
2. Ferrite changes and dissolves Cementite.
as a SOLID SOLUTION called AUSTENITE.
3. QUENCH.

AUSTENITE.

MARTENSITE. TROOSTITE. SORBITE.

STRENGTHENING TREATMENT.

Re-heat to temper.

MARTENSITE.	Hard and brittle.
TROOSTITE.	450F to 500F. Hard and tough.
SORBITE.	977F. strong and ductile.
PEARLITE.	Soft.

OUTLINE of HEAT TREATMENT.

THE MECHANICS OF SIMPLE MACHINE SHOP TOOLS

The illustrations in this lesson show some tools and devices used in the Machine Shop. On one side there are line diagrams (numbered), showing the mechanical principles involved, and on the other side the tool or device (lettered). Any practical man knows the importance of really understanding the fundamental laws of mechanics. It is not sufficient to understand the pure theories, but it is necessary to be able to recognize them in a practical environment and be able to make use of them. Many experienced men call such a knowledge "horse sense", and is certainly a very important but neglected phase of modern educational methods. A student therefore would be advised to extract his theories from concrete examples, as is shown in this lesson.

Diagrams A and B show two positions of a file being used to file a piece of work. If the left hand holds the point of the file and the right hand holds the handle, one can readily see that to keep the file in equilibrium to file the work square is no easy matter. The pressure on each end must change continually as the file moves, on account of the changing length of the pressure arms.

In Diagrams (1) and (2), pressure C is to pressure D, as the length B is to the length A. Stating this as an equation:

$$C \times A = B \times D$$

In Diagram (1). If the length A=2″ and length B 8″ and pressure at D 4 lbs., find pressure at C.

$$C = \frac{B \times D}{A} = \frac{8 \times 4}{2} = 16 \text{ lbs.}$$

In Diagram (2). If length A=6″ and length B 4″ and pressure D 4 lbs., find pressure at C.

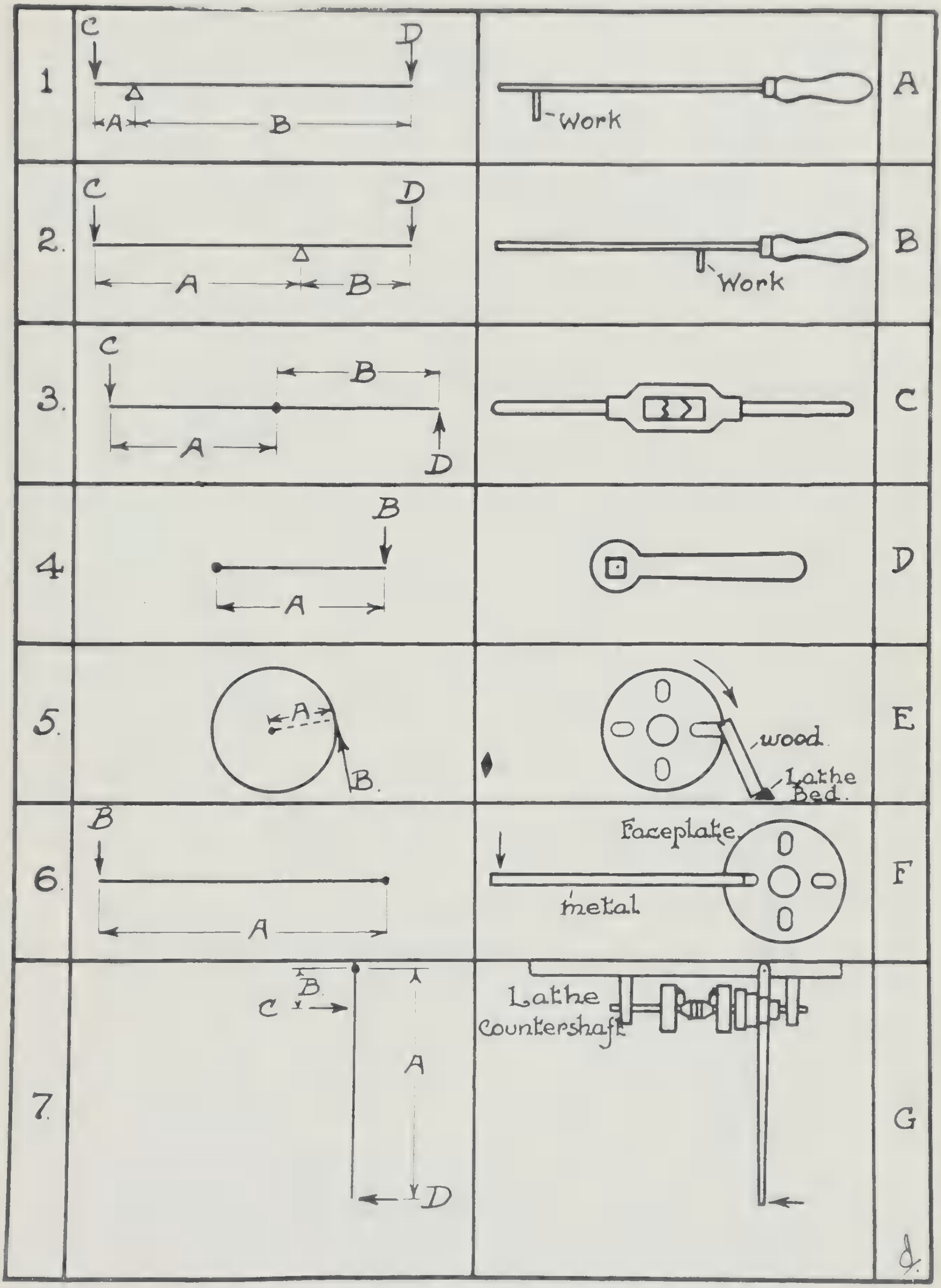
$$C = \frac{B \times D}{A} = \frac{4 \times 4}{6} = 2\tfrac{2}{3} \text{ lbs.}$$

The above problems show the varying pressures required for different positions of the file.

Diagram (C) illustrates a Tap Wrench which is used to turn taps into a hole when tapping a thread. The pressure from the left and right hand of the operator acts at equal distances from the centre of rotation.

In Diagram (3) both forces C and D help to turn the wrench, so that the turning moment equals $(C \times A) + (B \times D)$ or $D \times (A + B)$ or $C \times (A + B)$. This balanced action is known in mechanics as a "Couple."

Diagram (D) illustrates a common wrench. It can be seen from the line diagram (4) that the force B acts at a distance A from the centre of rotation to turn the wrench and consequently the nut.



1
C
D
A
B
Work
A
2.
C
D
A
B
Work
B
3.
C
B
A
D
C
4
B
A
D
5.
A
B.
wood
Lathe
Bed.
E
6.
B
A
Faceplate
metal
F
7.
B
C
A
D
Lathe
Countershaft
G

A x B = the turning moment.

If A = 6″ and B 10 lbs., turning moment = 60—pounds—inches.

If A = 8″ and B 10 lbs., turning moment = 80—pound—inches.

Therefore, the moment increases with the increased length of A.

Diagram (E) illustrates an application of turning moment when loosening the faceplate from a lathe spindle. The moment equals A x B, diagram (5). If this fails to loosen the faceplate, the length A may be increased as shown in diagram F and line diagram (6). Since A in diagram (6) is 4 times the length of A in diagram (5), the moment will be 4 times greater for the same pressure applied.

Diagram (G) shows the shipper pole of a lathe countershaft used to engage and disengage the clutch. From line diagram (7), the resisting moment equals C x B and the power moment D x A. If B = 2 feet and A = 8 feet and resistance C = 12 lbs., find power applied at D. C x B = D x A. Therefore, $D = \frac{C \times B}{A} = \frac{12 \times 2}{8} \quad \frac{24}{8} = 3$ lbs.

The ratio of movement will be in the same proportion, but inversely proportional. *Example.* If C moves 3″, how far does D move?

C movement : D movement : : Distance B : distance A.

Therefore C x A = D x B

$$D = \frac{C \times A}{B} = \frac{3'' \times 96''}{24''} = 12''$$

THE MECHANICS OF LATHE TOOLS

Successful use of the lathe and lathe tools, depends largely upon the operator's ability to recognize the mechanical principles involved in their use. Tools may be used in such a manner that their position causes undue strain on the tool itself, or causes strain on the machine and produces poor work. It is the purpose of this lesson to point out some of the outstanding errors noticed in the common use of the lathe.

The adjacent diagram sheet shows on the right hand side sketches of lathe tools in use and on the left hand side the mechanical diagram of such tools.

Forces acting on a lathe tool held in a tool post, (A). As the work rotates against the point of the tool, the pressure of the chip as it is removed tends to press the tool down. This force acts as a moment about the last point of support of the toolholder at (X). The toolholder is prevented from moving down under the chip pressure by the pressure of the set screw acting on the toolholder at the point Y, which offers a resisting moment about the last point of support X. Referring to the line diagram (1) of this tool, one can see that the power moment to move the tool equals the pressure of the chip multiplied by the distance (ZX). The resisting moment equals the pressure of the set screw multiplied by the distance (XY). By comparing tool (A) and tool (B) it is readily observed that the power moment in (B) is much greater than in tool (A) and the resistance moment is the same. Therefore position (A) is much better than position (B).

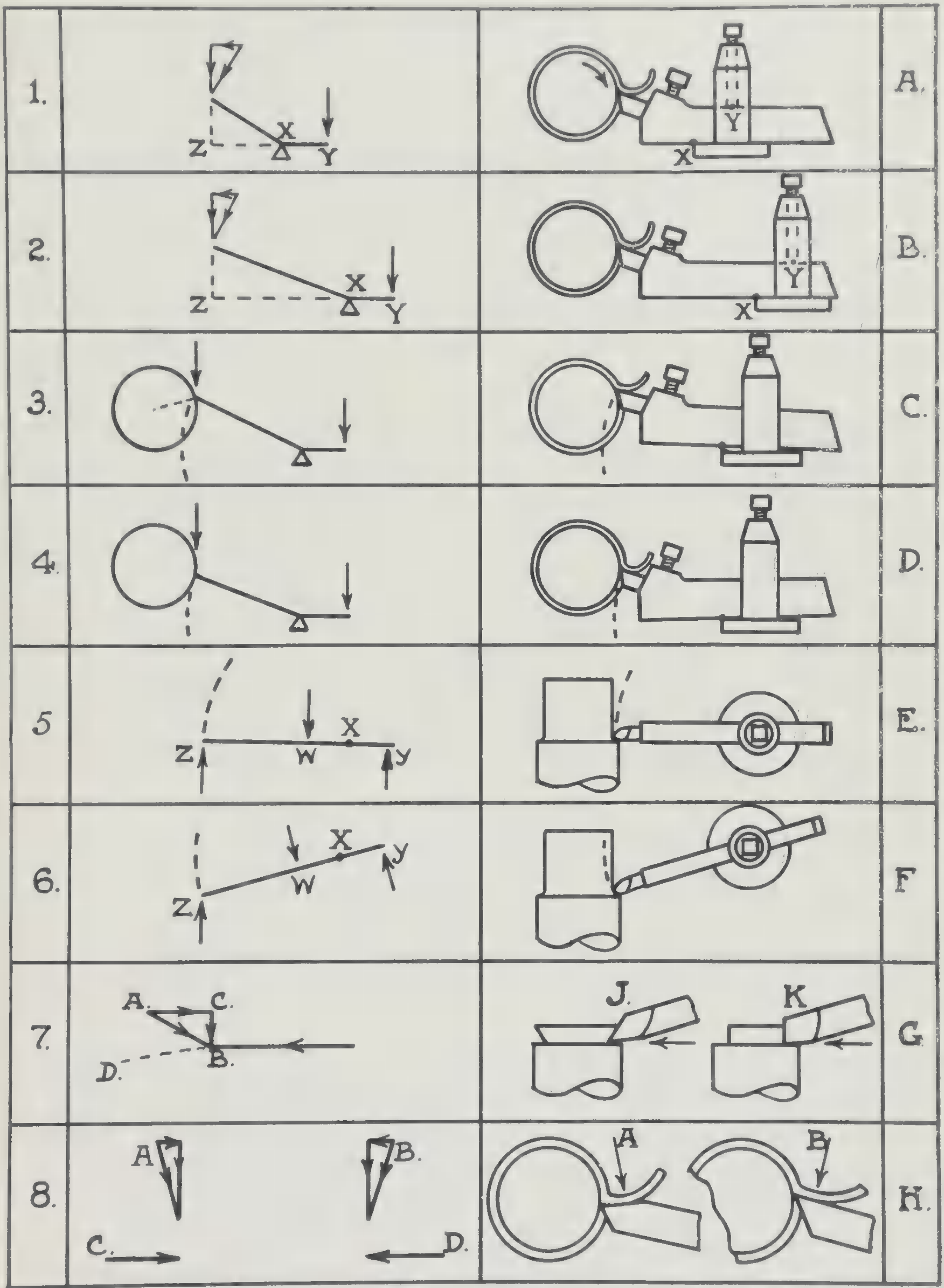
The effect of the height of the tool on the work. Tool (C) shows the correct height for turning, providing that the tool is rigid. If the tool moves down under chip pressure because it is not supported properly, the effect will be that it swings down into the work, as shown by the dotted line. If the point of the tool is set on centre, as at (D), under extreme pressure, it would swing down but away from the work and would not spoil the work by reducing the diameter as in (C). The position (D) is used in threading, because of the pinching action on the tool giving increased pressure. In position (C), this movement can be prevented by giving proper support to the tool as at (A).

The plan position of the tool in relation to the work. As the tool is fed along the work parallel to the axis, the pressure of the cutting tends to turn it around in the tool post with a moment equal to force (Z) multiplied by the distance (ZX). The pressure moment acts as a couple, as shown at WX, and if greater than the resisting moment on the base of the tool post the tool will turn. If the tool is placed as at (F), it will be

turned into the work and thus reduce its diameter and probably spoil it. If the tool is placed as at (E) and turns under pressure, it can only move away from the work and increase the turned diameter of the stock.

The triangle of forces and its application to lathe tools. Diagram (G) shows the plan of two tools squaring a piece of stock. The tool at (J) if moved at right angles to the axis of the work under the control of the cross feed would meet with the resistance of the work acting as a force in the direction (AB) in the line diagram. This resultant force has a component force (CB), which tends to push the tool into the work parallel with the axis. As a result of this, the tool instead of progressing at right angles to the axis is deflected, as shown by the dotted line (BD), and produces a concave end on the work. The tool shown in plan at (K) would not meet any deflecting force other than the resistance of the work acting at right angles to its cutting face and consequently would make the work square.

Effect of chip pressure on the front rake of a tool (H). As the chip strikes the top face of the tool the force acts at right angles to that face, as shown by the arrows (A) and (B). The tool turning brass, as at (A), has a negative front rake. The horizontal component of the force (A) tends to push the tool away from the work as shown in the diagram at (C). This action tends to offset the pulling in the action of brass on a turning tool. The tool turning machine steel, as at (B), shows the force acting at right angles to the top face of the tool by the arrow (B). The horizontal component of this force tends to push the tool into the work and helps to offset the resistance to penetration of the work, as shown in the line diagram at (D). It can be seen from the above examples that a knowledge of elementary mechanics and their influence upon lathe tools has a great deal to do with their effective operation.


1.
Z
X
Y
A.
Y
X
2.
Z
X
Y
B.
Y
X
3.
C.
4.
D.
5
Z
W
X
Y
E.
6.
X
W
Y
Z
F.
7.
A.
C.
B.
D.
J.
K.
G.
8.
A
B.
C.
D.
A
B
H.

MACHINE TOOL OPERATIONS

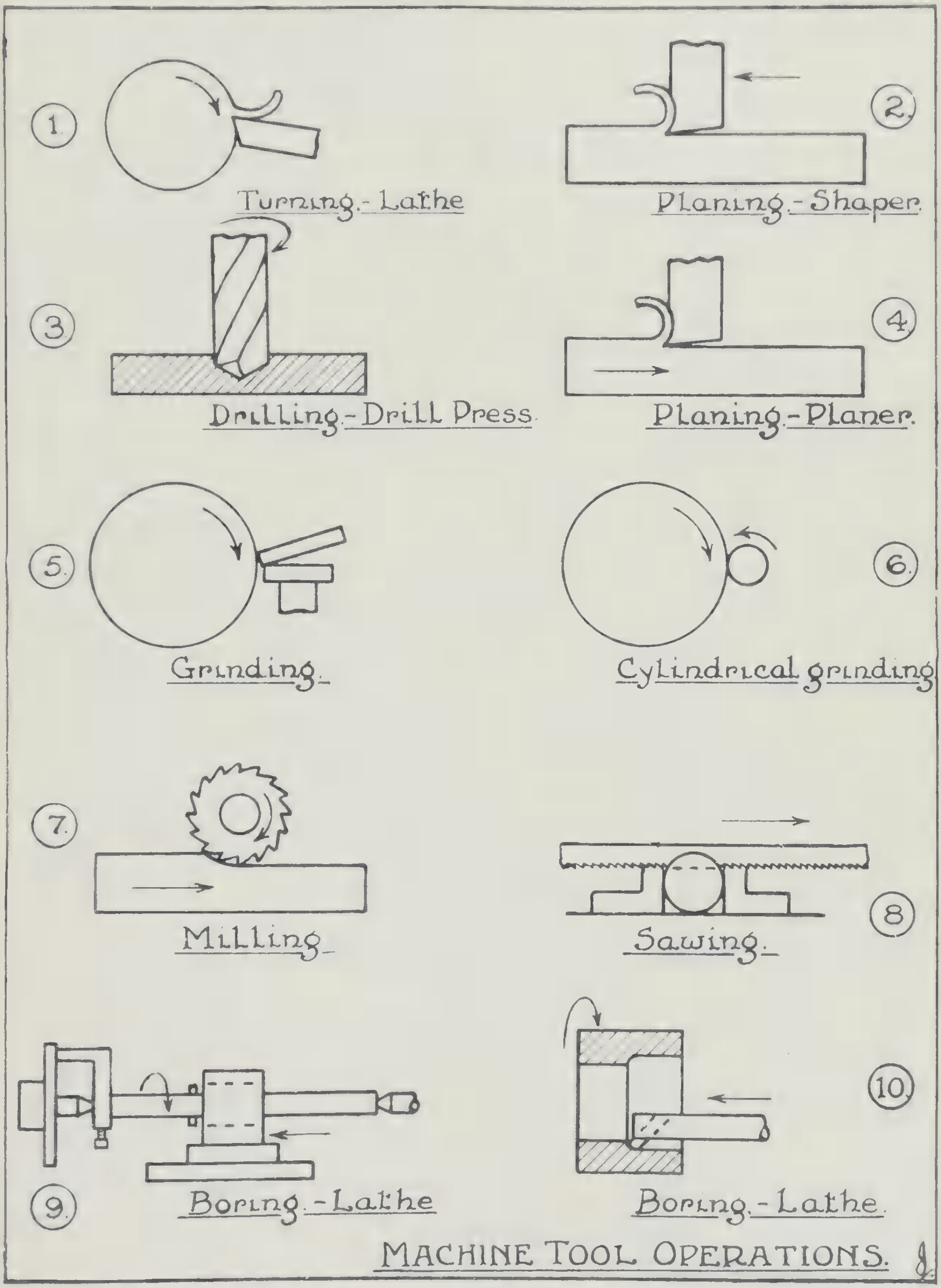
The diagrams shown in this lesson illustrate the various basic machine tool operations, so that by comparing and contrasting the features of each, one may realize the advantages or disadvantages of any particular method of removing or separating metal. One of the most important observations that will result from such an investigation of these fundamental processes is the fact that the speed of the removal of metal is so dependent upon the cutting speed the tool will stand. High speed steel gave an increase to cutting speeds over carbon tool steel with double the cutting speed and now we expect new and greater speeds, possibly four times that which can be obtained with high speed steel tools by the use of the new cemented tungsten carbide, which is a cobalt, carbon, tungsten alloy.

Diagram (1). Turning in the lathe. Here the work rotates past a stationary tool. The relation of the tool to the work and the angles of the tool are fully dealt with in lessons on tool grinding, page 109. The heat generated by cutting plays an important part in the operation, and must at all times be given consideration. It can be dissipated to some extent by the use of proper refrigerants, the commonest being a water solution with some animal fat to assist the cutting and sal soda to prevent the water from rusting the work. This provision for the absorption of the heat generated when cutting is necessary for all the operations illustrated here, with various compounds used for different metals. There are some exceptions, such as cast iron, which cuts better dry.

Diagram (2). Planing on the shaper. Here the work is held stationary and the tool cuts on the forward stroke and passes over the work without cutting on the return stroke. This method of metal removal differs from lathe work in that lathe turning is continuous, and planing is intermittent. The shaper is used for small work, primarily for producing flat surfaces.

Diagram (4). Planing in the planer. Here the tool is stationary and the work passes the tool. It is fastened to a table which passes the tool on the cutting stroke, then returns without cutting, ready for the next stroke. This method of producing flat surfaces is used on work too large for the shaper, although the tool operations are very similar.

Diagram (3). Drilling in the drill press. Usually two lip drills are used so that as the drill rotates two cutting edges remove metal at the same time. This operation is another example of continuous cutting. It is very important that the drill be properly ground to give a balanced


1
Turning.-Lathe
2
Planing.-Shaper.
3
Drilling.-Drill Press.
4
Planing.-Planer.
5
Grinding.
6
Cylindrical grinding
7
Milling.
8
Sawing.
9
Boring.-Lathe
10
Boring.-Lathe.
MACHINE TOOL OPERATIONS.

cutting action. The conical point of the tool tends to direct the cutting in its true location. See page 29 on drill grinding.

Diagram (5). Grinding metal on a grindstone. The grindstone is a multiple cutting tool of the continuous cutting type. The relief of each cutting abrasive grain is provided by the porosity of the wheel, which also helps to keep the cutting action cool. If the proper wheel is selected for a particular purpose, the size of the grains will be suitable and the grade of the wheel will provide that as soon as any grain ceases to cut, the extra pressure on its dull edges will break the bond that holds the grains together and uncover a new sharp grain which will cut efficiently.

Diagram (6). Cylindrical grinding. Here the grindstone and the work rotate to produce finished work of a very high calibre. The work produced in this way is much rounder than can be produced on the lathe, and it is possible in this way to finish work which has been hardened to very close limits of size.

Diagram (7). Milling. There are many methods of removing metal on the milling machine, but generally the cutter rotates while the work moves in a straight line past the rotating cutter. This is another example of continuous cutting, with a very big range of work with all sorts of cutting possibilities. It is in some ways similar to a grindstone because each cutting edge of the tool cuts off a small amount of metal as it rotates.

Diagram (8). Sawing. The reciprocating saw, like the shaper, is intermittent in its action. High speed steel blades have increased the cutting speed of power saws. For many years this tool lagged behind when compared to other tools, but now with high speed blades and positive feed machines, it is much improved. But even yet the operation of severing metal is a comparatively slow one. Circular saws and abrasive wheels are being used for the purpose of cutting off bar stock to advantage.

Diagram (9). Boring in the lathe (work not rotating). Here the cutter rotates on centres while the work is fastened to the carriage and is fed along to bore out the desired hole.

Diagram (10). Boring in the lathe (work rotating). Here the boring bar is fastened to the carriage through the tool post and is fed by it into the work while it rotates past the cutter. Boring produces holes that are round, much more so than those produced by drilling.

PROCESSES USED IN THE MANUFACTURE OF STEEL

The Bessemer Process. This process consists chiefly of removing nearly all the impurities from pig iron by blowing a current of air through the molten metal. It is essentially an oxidizing process, the molten iron is taken from the blast furnace, conveyed by ladle and poured into a pitcher-shaped vessel which is lined with fire brick called a "Bessemer Converter". When the gases coming from the open end of the converter show by a change in colour that the impurities and carbon are burnt out, a special iron rich in carbon, manganese and silicon is added to give the steel the desired composition. When this has thoroughly mixed with the metal the converter is tipped and the metal is poured into a ladle from which it is run into ingot moulds.

Bessemer steel is produced quickly and is used for the cheaper grades of steel, for rails and structural work, but for a better class of work the steel produced by the slower open hearth process is preferred.

The Open Hearth Process. In this process the impurities in pig iron are oxidized by adding iron oxide, and diluted by adding scrap steel. The charge is placed in the open hearth furnace which consists of a brick room with a low arched roof above the hearth. The charge is melted by a flame passing over it which is a mixed hot blast of air and gas.

Lime and other non-metallic substances are put in the furnace and melt forming a "slag" which floats on the surface of the metal and helps to refine it, and protect it from the direct action of the flame.

Open Hearth Steel is used for bridges, the better class of structural steel work, armour plate and also for conversion into high grade tool steel.

The Crucible Process. Crucible steel is produced by melting wrought iron or steel in a graphite crucible. The crucible is charged with equal quantities of the best puddled iron and scrap steel (usually ends trimmed from tool steel bars) or sufficient rich alloys and charcoal to make it meet the required analysis.

The crucible is lowered into a melting hole and surrounded by burning coke. When melted the contents are poured into metal molds forming ingots of steel.

This process is expensive for producing high grade steels and is being replaced by the electric furnace process.

The Electric Process. This kettle-shaped furnace is lined with fire brick and has a low domed-shaped roof through which the electrodes pass.

The front is provided with a pouring spout and the metal is charged into the furnace through side doors.

A very high current is sent through the electrodes which causes arcs to form between the lower ends of the electrodes and the metal below. The carbon is almost entirely burnt out and the proper amount is introduced to procure a high-grade carbon steel or an alloy steel. The steels produced by this process cover a great range and are of excellent quality.

The Cupola Furnace. This furnace is used for remelting iron or steel for casting into molds which have been prepared in the sand from wood or metal patterns. For cast iron a mixture of pig iron, coke and limestone is used and melted by means of an air blast similar to a blast furnace.

SUMMARY

Of the Qualities of Various Ingredients of Steel

IRON	The basis of steel.
CARBON	The determinative.
SULPHUR	A strength sapper.
PHOSPHOROUS	A weak link.
OXYGEN	A strength destroyer.
MANGANESE	To give strength.
NICKEL	For strength and toughness.
TUNGSTEN	Hardener and heat resister.
CHROMIUM	For resisting shocks.
VANADIUM	Purifier and fatigue resister.
SILICON	Impurity and hardener.
TITANIUM	Removes nitrogen and oxygen.
MOLYBDENUM	Hardener and heat resister.
ALUMINUM	Kills or de-oxidizes steel

Note:

CAST IRON is Crystalline in structure.
WROUGHT IRON is Fibrous in structure.
TOOL STEEL is Granular in structure.

Outline of Processes

IRON ORE

PROCESS

BLAST FURNACE

PIG IRON

CONVERSION PROCESSES	REMELTING PROCESSES
1 BESSEMER CONVERTER Rails Plates Structural shapes	1 CUPOLA FURNACE White Cast Iron
2 BASIC OPEN HEARTH FURNACE Rails Plates Structural shapes	2 ANNEALING FURNACE Malleable Cast Iron
3 ACID OPEN HEARTH FURNACE High grade steel	3 CUPOLA FURNACE Gray Cast Iron. Castings for Machinery
4 ELECTRIC FURNACE High grade steel Tool steel	
5 PUDDLING FURNACE Wrought Iron	
6 REMELTED IN CRUCIBLES Crucible steel Tool steel	

CLASSIFICATION OF STEEL

(A) Carbon Steels

NAME	CONTENT	FEATURE	USE
VERY MILD STEEL	Less than .10% carbon	Very soft	Rivets, pipe, chain, etc.
MACHINE STEEL MILD STEEL	.10% to .25% carbon	Soft, suitable for case-hardening	Structural steel work, case hardened parts, bolts, etc.
MACHINE STEEL	.25% to .60% carbon	Harder than mild steel. Can be partially hardened	Shafting car axles, automobile parts
TOOL STEEL	.60% to 1.50% carbon	Can be hardened and tempered	For making tools, springs, files, etc.

(B) Alloy Steels

CHROME STEEL (Low)	Carbon .9% Chromium .50%	Fine grain, Very tough	Chisels, hammers, Axes
CHROME STEEL (High)	Carbon 1.00% Chromium 1.25%	Strong and hard	Balls and races
CHROME STAINLESS STEEL	Carbon 1.00% Chromium 8% to 15%	Resists corrosion	Knives and exhaust valves
NICKEL STEEL	.2% to .5% nickel Carbon 0.25%	Hard, strong, elastic, ductile	Automotive forgings, armour plate, etc.
CHROME NICKEL STEEL	Low. 1.5%N, .60%C Med. 2.5%N, 1.00%C High 3.50%N, 1.5%C	Strong and tough very suitable for case hardening	Automobile parts, axles, gears, crankshafts, die blocks, etc.
CHROME-VANADIUM STEEL	.30% carbon Chromium .1% Vanadium .20%	Toughness and shock resisting qualities	Structural parts
CHROME-VANADIUM STEEL	Carbon .50% Chromium 1% Vanadium .20%	Toughness and shock resisting qualities	Springs and gears
MANGANESE STEEL	.12% to .14% Manganese. Carbon 1.5%	Suddenly cooled it is ductile, slowly cooled it is brittle, opposite to high carbon steel. Resists wear, great hardness	Switch points, dredge bucket teeth, safes

SILICO MANGANESE STEEL	Silicon .2% Manganese .60%—.80% Carbon .50%—.60%	Suitable for springs	Leaf springs for railroad and automotive work
HIGH SPEED STEEL	Carbon .60%—.80% Chromium 3.50%—4.00% Tungsten 12.00%—18.00% Vanadium .75%—1.50%	Resists heat	Cutters and tools
NON-SHRINKING OIL-HARDENING STEEL	Carbon .80%—.90% Vanadium .20%—.25% Manganese 1.40%—1.60%	Low rate of expansion and contraction in hardening	Gauges, punches, dies, taps, etc.

(C) Non-steel Alloy

STELLITE	Cobalt 60% Chromium 1% Molybdenum, 23% No Iron	Cast, very brittle, cuts at red heat	Tool bits

(D) Iron

CAST IRON	3% to 5% carbon (in graphitic form)	Hard skin, soft interior, brittle, high compressive strength and cheap, crystalline structure	Beds of machines and shaped parts which do not have to resist a great strain
WROUGHT IRON or PUDDLED IRON	Very little carbon 0.05% to 0.3% (in cementite form)	Fibrous structure and tough, fairly soft and bends easily	

(F)

CEMENTED TUNGSTEN CARBIDE	Carbon, Tungsten, Cobalt	Extremely hard, very expensive	For cutting tools

SHOP SCIENCE

1. If gear and pinion blanks of a certain approximate diameter were required of a certain definite ratio, what points would you consider as to whether large or small teeth should be cut on them?
2. What do you understand by: (a) Diametral pitch, (b) Circular pitch?
3. Sketch a gear and pinion showing only, pitch circle, addendum, whole depth, module.
4. Show by means of a sketch, how the base circle is obtained, make an enlarged sketch of 2 teeth showing the different parts of the tooth face with the names of the parts.
5. Show approximately by means of a freehand sketch the difference in shape between a cutter used for a small pinion and one used for a large gear of the same diametral pitch.
6. Sketch freehand 3 involute curves as generated from the base circle of a gear. What do you know about the distance between these curves if measured at a tangent to the base circle? What use is made of this fact?
7. If you are given a pinion with the number of teeth and the outside diameter: Design a gear to mate with it. Ratio 2 to 1 with the pinion.

   Find:— (a) Number of teeth in gear, (b) Outside diameter of gear, (c) Diametral pitch, (d) Circular pitch, (e) Thickness of tooth, (f) Addendum, (g) Clearance, (h) Whole depth, (i) Pitch diameter, (j) Centre distance.
8. State 3 methods of finding Diametral pitch of a gear.
9. State 3 methods of finding the pitch diameter of a gear.
10. State 3 methods of finding the outside diameter of a gear.
11. If a gear were handed to you, how would you find the diametral pitch of it?
12. Make a vernier with 2 pieces of paper, the main scale having 10 divisions to the inch, and the vernier reading in hundredths of an inch.
13. Explain how a micrometer is designed to measure correctly to 1/10,000"?
15. Define Annealing, Normalizing.
16. Define Hardening, Tempering.
17. Define Carburizing, Cyaniding, Case-hardening.
18. What carbon content would you select for the following uses:— Cold chisels, Drills, Taps, Turning tools, Hammers?
19. What is the approximate Hardening Heat of the following Carbon Content tool steels? (1) 70 point carbon, (2) 80 point carbon, (3) 90 point carbon, (4) 100 point carbon?

20. Should steel be quenched outright in cold water? Should the steel be left in air without tempering? Give reasons for your statements.
21. What approximate temperatures would you use in tempering carbon tool steel: (a) To relieve strains only? (b) To relieve strains and reduce brittleness? (c) To relieve strains and to toughen? What tempering medium would you use for each?
22. What approximate temperature would you use when tempering lathe cutting tools, Scrapers, Hammer faces, Hand taps, Milling cutters, Twist drills, Cold chisels, Screw drivers?
23. Make a line diagram to illustrate the Decalescent and Recalescent points in the heating and cooling of steel. Indicate on the diagram the hardening heat of the steel.
24. What apparatus would you use to prove that a piece of steel contracts while heating when at the Decalescent point, and expands when cooling at the Recalescent point?
25. Explain the phenomena of the "Critical points" or "Transformation points" when heating tool steel.
26. Make line diagrams to illustrate Annealing, Normalizing.
27. Make a line diagram to illustrate Hardening and Tempering. Show Heating time, Cooling time, Heating and Cooling temperatures, time for holding temperature approximately. Draw lines to show room temperature and Hardening Heat or Critical period.
28. Why is it necessary to Anneal tool steel? What qualities does it give to the steel?
29. What kind of work do your Normalize? What is the purpose of this operation?
30. What is the effect of: (a) Prolonged heating of tool steel? (b) Rapid heating of tool steel? (c) Impinging heat on one portion of tool steel?
31. State some reason for cracks occurring when hardening tool steels.
32. Discuss the heating of tool steel which has: (a) Large and small masses, (b) Work with holes drilled in it.
33. Why does the correct shape of work help to prevent hardening cracks?
34. How would you harden a milling cutter to leave the cutting edges file hard, and the centre tough?
35. Why should work not be left exposed to the air after quenching to harden? Why should the work be tempered immediately after hardening?
36. How should the following work be quenched? (a) With a large and small masses, (b) Flat work. What are the effects of improper quenching?
37. What is the effect of: (a) Overheating steel? (b) Not heating steel sufficiently?

38. What are the advantages and disadvantages of the following equipment for use in Hardening Steel? (a) A Forge fire, (b) A Gas forge, (c) A Muffle furnace (gas or oil), (d) A Lead bath, (e) A Salt bath, (f) An Electric furnace.

39. Describe a Thermocouple and a Pyrometer for measuring the temperature of a furnace.

40. How would you test the Hardening Heat of Steel without a temperature testing instrument?

41. Discuss the advantages and disadvantages of tempering (a) By colours, (b) with thermometer?

42. What is the effect of hammering steel: (a) When hot, (b) When cold?

43. Why is steel often heated in sealed boxes instead of heating in a fire or furnace?

44. Define the following conditions of steel: Austenite, Martensite, Troostite, Sorbite, Pearlite.

45. What effect does the quenching rate have when withdrawing heat from steel when quenching to harden the steel?

46. (a) What condition is steel in when quenched in water from the proper hardening heat? (b) What defects does the steel have in this condition? (c) Why is it advisable to test the steel with a file after quenching?

47. What do fractures indicate to an operator when testing the effect of the hardening operation?

48. What are the chief reasons for tempering steel after hardening? What conditions may be obtained by this process?

49. Make line diagrams showing the mechanics involved when using the following tools: (a) A File, (b) A Tap wrench, (c) An ordinary wrench, (d) Removing a faceplate of a lathe with a lever.

50. Show by means of a line diagram the mechanics of the following lathe tool positions: (a) A tool turning metal, (b) A tool cutting above the centre of the work and one on the centre of the work.

51. Explain why a tool holder should be set if possible in a position to to turn away from the work, not into the work, if the tool happens to move.

52. Show the effect of the chip pressure on a lathe tool: (a) With front rake, (b) With a negative front rake.

53. Why should a tool never be allowed to cut so that pressure occurs on two sides of an acute angle at its cutting edges, when seen in profile?

54. Make simple sketches showing the following tool operations. Turning. Boring with work rotating. Boring with tool rotating and work stationary.

55. Make simple sketches showing the following operations:—Planing on the Shaper. Planing in the planer. Milling with plain mill. Cylindrical grinding.

56. Explain briefly the properties and uses of the following metals: Wrought Iron, Cast Iron, Cast Steel, Machine Steel.

57. Describe the features and uses of at least four alloy steels.

58. Describe the features and uses of the following cutting mediums. Carbon tool steel, High Speed Steel, Stellite, Tungsten Carbide.

TEMPERATURES FOR FORGING AND HARDENING.

Temper.	Carbon.	Heating for Forging.			Heating for Hardening.		
		Maximum Colour.	Maximum Temperature.		Maximum Colour.	Maximum Temperature.	
			Centigrade.	Fahr.		Centigrade.	Fahr.
1	1½%	Bright Cherry Red	825°	*1517°*	Blood or low Cherry Red	730°	*1346°*
2	1¼%	,,	825°	*1517°*	,,	730°	*1346°*
3	1⅛%	Full Red	850°	*1562°*	Medium Cherry Red	750°	*1382°*
4	1%	,,	850°	*1562°*	,,	760°	*1400°*
5	⅞%	,,	850°	*1562°*	Cherry Red	780°	*1436°*
6	¾%	,,	850°	*1562°*	,,	780°	*1436°*

TEMPERING

Temper Colour.	Centigrade.	Fahrenheit.
Light Straw	220°	*428°*
Straw	230°	*446°*
Dark Straw	240°	*464°*
Brown Yellow	255°	*491°*
Red Brown	265°	*509°*
Purple	275°	*527°*
Purple Blue	285°	*545°*
Full Blue	295°	*563°*
Pale Blue	310°	*590°*
Grey	330°	*626°*

Tapers and Angles

Taper per Foot	Included ∢			With Center Line ∢			Taper Per Inch	Taper per Inch from Center Line
	Deg.	Min.	Sec.	Deg.	Min.	Sec.		
⅛	0	35	48	0	17	54	.010416	005203
3/16	0	53	44	0	26	52	015625	.007812
¼	1	11	36	0	35	48	020833	.010416
5/16	1	29	30	0	44	45	026042	.013021
⅜	1	47	24	0	53	42	031250	.015625
7/16	2	5	18	1	2	39	036458	.018229
½	2	23	10	1	11	35	.041667	.020833
9/16	2	41	4	1	20	32	.046875	.023438
⅝	2	59	42	1	29	51	.052084	.026042
11/16	3	16	54	1	38	27	057292	.028646
¾	3	34	44	1	47	22	062500	.031250
13/16	3	52	38	1	56	19	067708	033854
⅞	4	10	32	2	5	16	072917	.036456
15/16	4	28	24	2	14	12	078125	.039063
1	4	46	18	2	23	9	.083330	.041667
1¼	5	57	48	2	58	54	.104666	.052084
1½	7	9	10	3	34	35	125000	.062500
1¾	8	20	26	4	10	13	145833	072917
2	9	31	36	4	45	48	166666	083332
2½	11	53	36	5	56	48	.208333	104166
3	14	15	0	7	7	30	250000	125000
3½	16	35	40	8	17	50	291666	.145833
4	18	55	28	9	27	44	333333	166666
4½	21	14	2	10	37	1	.375000	.187500
5	23	32	12	11	46	6	416666	.208333
6	28	4	2	14	2	1	.500000	.250000

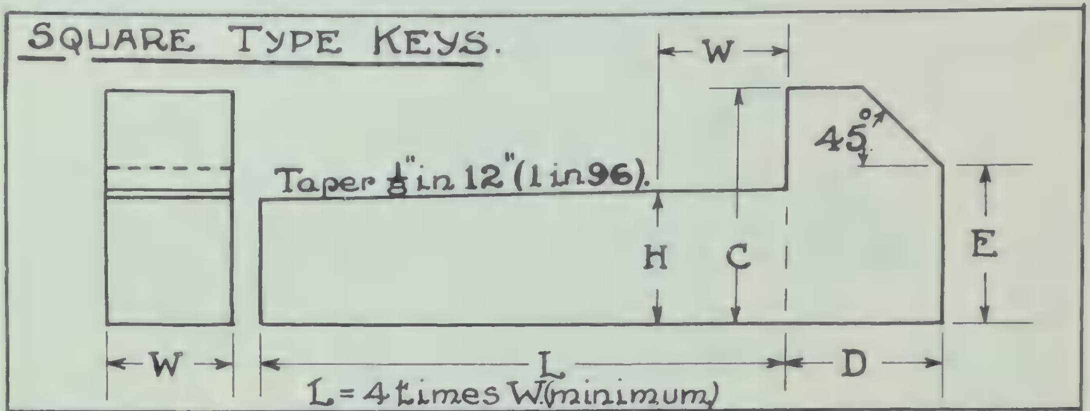
HARDNESS VALUES of METALS for the SHORE SCLEROSCOPE.						
NAME OF METAL	ANNEALED OR CAST.		COLD WORKED		CHILLED OR HARDENED.	
LEAD	2	4	3	7		
GOLD, 24 TO 14 carat	5	25	24	70		
SILVER	$6\frac{1}{2}$	14	20	37		
COPPER	6	8	14	20		
ZINC	8	10	18	20		
BABBITT METAL	4	9				
TIN	8	9	12	14		
BISMUTH	8	9				
BRASS	7	35	20	45		
PLATINUM	10	15	17	30		
BRONZE (PHOSPHOR)	12	21	25	40		
BRONZE (MANGANESE)	16	21	25	40		
IRON, WROUGHT	16	18	25	30		
NICKEL, WROUGHT	17	19	35	40		
MILD STEEL (·05 to ·15 carbon)	18	25	30	40		
IRON, GRAY (SAND CAST)	25	45				
TUNGSTEN	60	70			60	70
IRON, GRAY, CHILLED.					50	90
STEEL, TOOL (1% carbon)	30	35	40	50	90	110
STEEL, TOOL (1·65% ")	38	45			90	110
STEEL, VANADIUM	30	50	50	60	50	110
STEEL, CHROME NICKEL	35	50	40	60	60	105
STEEL, NICKEL.	25	30	35	45	50	90
STEEL, HIGH SPEED.	30	45	40	60	70	105

— HARDNESS SCALES —
COMPARED

METAL	Sclerometer	Scleroscope	Brinell
LEAD	1.0	1.0	1.0
TIN	2.5	3.0	2.5
ZINC	6.0	7.0	7.5
COPPER, SOFT	8.0	8.0	....
COPPER, HARD	....	12.0	12.0
SOFTEST IRON	15.0	....	14.5
MILD STEEL	21.0	22.0	16-24
SOFT CAST IRON	21-24	24.0	24
RAIL STEEL	24.0	27.0	26-35
HARD CAST IRON	36.0	40.0	35.0
HARD WHITE IRON	72.0	70.0	75.0
HARDENED STEEL	....	95.0	93.0

8.

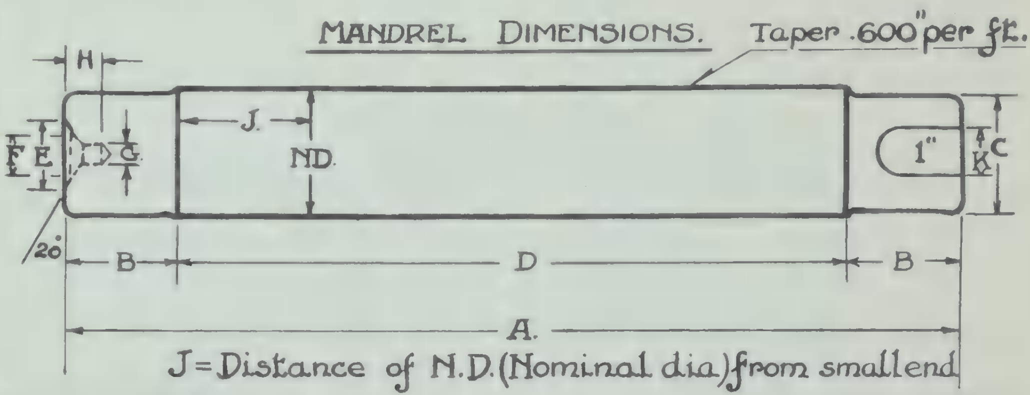
SQUARE TYPE KEYS.



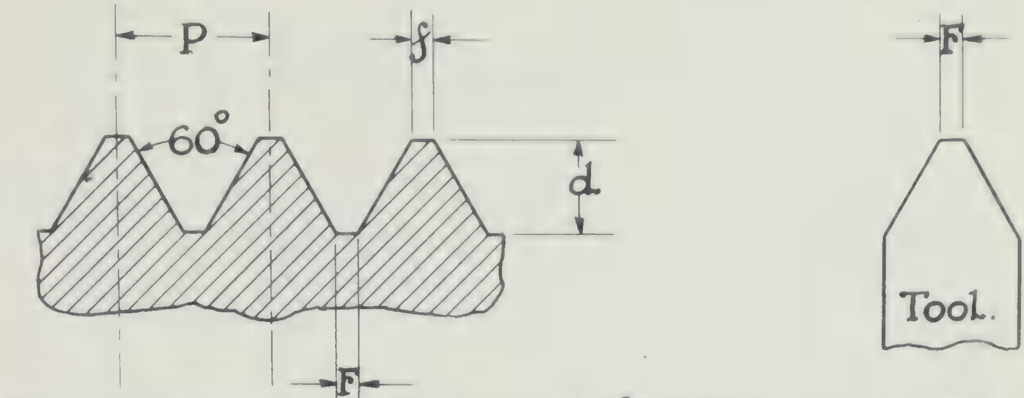
Diameter of Shafts. (Inclusive)	Key		Gib Head			Tolerances on Keys.	
	Maximum Width. W.	Minimum Height H.	Height C.	Length D.	Height E.	Width (Minus)	Height. (Plus)
1/2" to 9/16"	1/8"	1/8"	1/4"	7/32"	5/32"	0.0020	0.0020
5/8" " 7/8"	3/16	3/16	5/16	9/32	7/32	0.0020	0.0020
15/16" " 1 1/4"	1/4	1/4	7/16	11/32	11/32	0.0020	0.0020
1 5/16" " 1 3/4"	3/8	3/8	11/16	15/32	15/32	0.0020	0.0020
1 13/16" " 2 1/4"	1/2	1/2	7/8	19/32	5/8	0.0025	0.0025
2 5/16" " 2 3/4	5/8	5/8	1 1/16	23/32	3/4	0.0025	0.0025
2 7/8" " 3 1/4"	3/4	3/4	1 1/4	7/8	7/8	0.0025	0.0025
3 3/8" " 3 3/4"	7/8	7/8	1 1/2	1	1	0.0030	0.0030
3 7/8" " 4 1/2"	1	1	1 3/4	1 3/16	1 3/16	0.0030	0.0030
4 3/4" " 5 1/2"	1 1/4	1 1/4	2	1 7/16	1 7/16	0.0030	0.0030
5 3/4" " 6"	1 1/2	1 1/2	2 1/2	1 3/4	1 3/4	0.0030	0.0030

INVOLUTE CUTTERS	
No. on Cutter	Number of Teeth
1	135 to Rack
1½	80 to 134
2	55 to 134
2½	42 to 54
3	35 to 54
3½	30 to 34
4	26 to 34
4½	23 to 25
5	21 to 25
5½	19 to 20
6	17 to 20
6½	15 to 16
7	14 to 16
7½	13
8	12 to 13

EPICYCLOIDAL CUTTERS	
Letter on Cutter	Number of Teeth
A	12
B	13
C	14
D	15
E	16
F	17
G	18
H	19
I	20
J	21 to 22
K	23 to 24
L	25 to 26
M	27 to 29
N	30 to 33
O	34 to 37
P	38 to 42
Q	43 to 49
R	50 to 59
S	60 to 74
T	75 to 99
U	100 to 149
V	150 to 249
W	250 or more
X	Rack



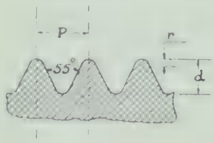
Nominal Diameter	Total Length	Length of Ends	Diameter of Ends	Length of Taper	Diameter of Recess	Diameter of Countersink	Drill Size	Width of Flat	Depth of Drilled Hole	N.D. from Small End
N.D.	A	B	C	D	E	F	G	K	H	J
$\frac{3}{8}$	$4\frac{1}{4}$	$\frac{1}{2}$	$\frac{11}{32}$	$3\frac{1}{4}$	$\frac{1}{4}$	$\frac{5}{32}$	$\frac{1}{16}$	$\frac{7}{32}$	$\frac{1}{4}$	$\frac{11}{16}$
$\frac{7}{16}$	$4\frac{1}{2}$	$\frac{1}{2}$	$\frac{13}{32}$	$3\frac{1}{2}$	$\frac{1}{4}$	$\frac{5}{32}$	$\frac{1}{16}$	$\frac{1}{4}$	$\frac{1}{4}$	$\frac{3}{4}$
$\frac{1}{2}$	5	$\frac{9}{16}$	$\frac{15}{32}$	$3\frac{7}{8}$	$\frac{5}{16}$	$\frac{3}{16}$	$\frac{5}{64}$	$\frac{9}{32}$	$\frac{9}{32}$	$\frac{13}{16}$
$\frac{9}{16}$	$5\frac{1}{4}$	$\frac{9}{16}$	$\frac{17}{32}$	$4\frac{1}{8}$	$\frac{5}{16}$	$\frac{3}{16}$	$\frac{5}{64}$	$\frac{9}{32}$	$\frac{9}{32}$	$\frac{7}{8}$
$\frac{5}{8}$	$5\frac{1}{2}$	$\frac{5}{8}$	$\frac{19}{32}$	$4\frac{1}{4}$	$\frac{13}{32}$	$\frac{7}{32}$	$\frac{5}{64}$	$\frac{9}{32}$	$\frac{5}{16}$	$\frac{15}{16}$
$\frac{11}{16}$	$5\frac{3}{4}$	$\frac{5}{8}$	$\frac{21}{32}$	$4\frac{1}{2}$	$\frac{3}{8}$	$\frac{7}{32}$	$\frac{5}{64}$	$\frac{5}{16}$	$\frac{5}{16}$	1
$\frac{3}{4}$	6	$\frac{3}{4}$	$\frac{23}{32}$	$4\frac{1}{2}$	$\frac{7}{16}$	$\frac{1}{4}$	$\frac{3}{32}$	$\frac{5}{16}$	$\frac{3}{8}$	$1\frac{1}{16}$
$\frac{7}{8}$	$6\frac{1}{2}$	$\frac{13}{16}$	$\frac{29}{32}$	$4\frac{7}{8}$	$\frac{17}{32}$	$\frac{9}{32}$	$\frac{3}{32}$	$\frac{3}{8}$	$\frac{13}{32}$	$1\frac{1}{8}$
1	7	$\frac{7}{8}$	$\frac{31}{32}$	$5\frac{1}{4}$	$\frac{9}{16}$	$\frac{5}{16}$	$\frac{3}{32}$	$\frac{3}{8}$	$\frac{7}{16}$	$1\frac{3}{16}$
$1\frac{1}{8}$	$7\frac{1}{2}$	1	$1\frac{1}{16}$	$5\frac{1}{2}$	$\frac{9}{16}$	$\frac{5}{16}$	$\frac{7}{64}$	$\frac{7}{16}$	$\frac{7}{16}$	$1\frac{5}{16}$
$1\frac{1}{4}$	8	1	$1\frac{3}{16}$	6	$\frac{5}{8}$	$\frac{11}{32}$	$\frac{7}{64}$	$\frac{1}{2}$	$\frac{15}{32}$	$1\frac{3}{8}$
$1\frac{3}{8}$	$8\frac{1}{2}$	$1\frac{1}{8}$	$1\frac{5}{16}$	$6\frac{1}{4}$	$\frac{11}{16}$	$\frac{3}{8}$	$\frac{7}{64}$	$\frac{1}{2}$	$\frac{1}{2}$	$1\frac{1}{2}$
$1\frac{1}{2}$	9	$1\frac{1}{8}$	$1\frac{7}{16}$	$6\frac{3}{4}$	$\frac{3}{4}$	$\frac{3}{8}$	$\frac{1}{8}$	$\frac{9}{16}$	$\frac{1}{2}$	$1\frac{9}{16}$
$1\frac{5}{8}$	$9\frac{1}{2}$	$1\frac{3}{16}$	$1\frac{9}{16}$	$7\frac{1}{8}$	$\frac{13}{16}$	$\frac{13}{32}$	$\frac{1}{8}$	$\frac{5}{8}$	$\frac{17}{32}$	$1\frac{11}{16}$
$1\frac{3}{4}$	10	$1\frac{3}{16}$	$1\frac{11}{16}$	$7\frac{5}{8}$	$\frac{7}{8}$	$\frac{13}{32}$	$\frac{1}{8}$	$\frac{5}{8}$	$\frac{9}{16}$	$1\frac{3}{4}$
$1\frac{7}{8}$	$10\frac{1}{2}$	$1\frac{1}{4}$	$1\frac{13}{16}$	8	$\frac{15}{16}$	$\frac{7}{16}$	$\frac{1}{8}$	$\frac{11}{16}$	$\frac{19}{32}$	$1\frac{7}{8}$



NOTE TAP DRILL SIZES – 75% DEPTH THREAD

U.S. Standard Thread			Size	S.A.E. Standard Thread		
Double Depth	Tap Drill Size	Number of Threads		Number of Threads	Tap Drill Size	Double Depth
.06495	$\frac{13}{64}''$	20	$\frac{1}{4}''$	28	$\frac{7}{32}''$	.04639
.07216	$\frac{1}{4}''$	18	$\frac{5}{16}''$	24	$\frac{9}{32}''$	.05412
.08118	$\frac{5}{16}''$	16	$\frac{3}{8}''$	24	$\frac{21}{64}''$	.05412
.09278	$\frac{3}{8}''$	14	$\frac{7}{16}''$	20	$\frac{25}{64}''$	.06495
.09992	$\frac{27}{64}''$	13	$\frac{1}{2}''$	20	$\frac{29}{64}''$	.06495
.10825	$\frac{31}{64}''$	12	$\frac{9}{16}''$	18	$\frac{33}{64}''$	.07216
.11809	$\frac{35}{64}''$	11	$\frac{5}{8}''$	18	$\frac{37}{64}''$	.07216
.11809	$\frac{39}{64}''$	11	$\frac{11}{16}''$	16	$\frac{5}{8}''$	.08118
.12990	$\frac{21}{32}''$	10	$\frac{3}{4}''$	16	$\frac{11}{16}''$	.08118
.14433	$\frac{49}{64}''$	9	$\frac{7}{8}''$	14 18	$\frac{13}{16}''$ $\frac{53}{64}''$	.09278 .07216
.16237	$\frac{7}{8}''$	8	$1''$	14	$\frac{15}{16}''$	.09278
.18557	$\frac{63}{64}''$	7	$1\frac{1}{8}''$	12	$1\frac{3}{64}''$	.10825
.18557	$1\frac{7}{64}''$	7	$1\frac{1}{4}''$	12	$1\frac{11}{64}''$	.10825
.21650	$1\frac{7}{32}''$	6	$1\frac{3}{8}''$	12	$1\frac{19}{64}''$	.10825
.21650	$1\frac{11}{32}''$	6	$1\frac{1}{2}''$	12	$1\frac{27}{64}''$	.10825

Whitworth Standard Threads

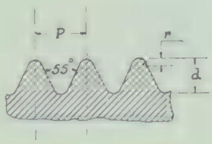


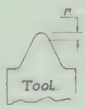
$$\text{Formula}\begin{cases} p = \text{pitch} = \dfrac{1}{\text{No. thds. per in}} \\ d = \text{depth} = p \times .64033 \\ r = \text{radius} = p \times .1373 \end{cases}$$

Diam., Inches	No. Threads per Inch.	Diam. Inches.	No. Threads per Inch.	Diam., Inches	No. Threads per Inch.	Diam., Inches	No. Threads per Inch.
1/4	20	7/8	9	2	4 1/2	3 1/4	3 1/4
5/16	18	15/16	9	2 1/8	4 1/2	3 3/8	3 1/4
3/8	16	1	8	2 1/4	4	3 1/2	3 1/4
7/16	14	1 1/8	7	2 3/8	4	3 5/8	3 1/4
1/2	12	1 1/4	7	2 1/2	4	3 3/4	3
9/16	12	1 3/8	6	2 5/8	4	3 7/8	3
5/8	11	1 1/2	6	2 3/4	3 1/2	4	3
11/16	11	1 5/8	5	2 7/8	3 1/2		
3/4	10	1 3/4	5	3	3 1/2		
13/16	10	1 7/8	4 1/2	3 1/8	3 1/2		

British Standard Fine Threads

(Formula same as Whitworth Standard)

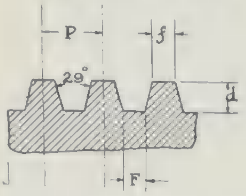




$$\text{Formula}\begin{cases} p = \text{pitch} = \dfrac{1}{\text{No. thds. per in.}} \\ d = \text{depth} = p \times .64033 \\ r = \text{radius} = p \times .1373 \end{cases}$$

Diameter, Inches.	Number of Threads per Inch	Diameter, Inches.	Number of Threads per Inch.
1/4	26	3/4	12
9/32	26	13/16	12
5/16	22	7/8	11
3/8	20	1	10
7/16	18	1 1/8	9
1/2	16	1 1/4	9
9/16	16	1 3/8	8
5/8	14	1 1/2	8
11/16	14		

Acme Standard Screw Thread.



$$p = \text{pitch} = \frac{1}{\text{No. thds. per in.}}$$

$$d = \text{depth} = \tfrac{3}{8}p \times .010$$

$$f = \text{flat on top of thread} = p \times .3707$$

$$f = \text{flat on bottom of thread} = p \times .3707 - .0052$$

Pitch.	No. of Threads per Inch.	Depth of Thread.	Width at Top of Thread.	Width at Bottom of Thread.	Space at Top of Thread.	Thickness at Root of Thread.
2	1/2	1.0100	.7414	.7362	1.2586	1.2637
1 7/8	8/15	.9475	.6950	.6897	1.1799	1.1850
1 3/4	4/7	.8859	.6487	.6435	1.1012	1.1064
1 5/8	8/13	.8225	.6025	.5973	1.0226	1.0277
1 1/2	2/3	.7600	.5560	.5508	.9439	.9491
1 7/16	16/23	.7287	.5329	.5277	.9046	.9097
1 3/8	8/11	.6975	.5097	.5045	.8652	.8704
1 5/16	16/21	.6662	.4865	.4813	.8259	.8311
1 1/4	4/5	.6350	.4633	.4581	.7866	.7918
1 3/16	16/19	.6037	.4402	.4350	.7472	.7525
1 1/8	8/9	.5725	.4170	.4118	.7079	.7131
1 1/16	16/17	.5412	.3938	.3886	.6686	.6739
1	1	.5100	.3707	.3655	.6293	.6345
15/16	1 1/15	.4787	.3476	.3424	.5898	.5850
7/8	1 1/7	.4475	.3243	.3191	.5506	.5558
13/16	1 3/13	.4162	.3012	.2960	.5112	.5164
3/4	1 1/3	.3850	.2780	.2728	.4720	.4772
11/16	1 5/11	.3537	.2548	.2496	.4327	.4379
2/3	1 1/2	.3433	.2471	.2419	.4194	.4246
5/8	1 3/5	.3225	.2316	.2264	.3934	.3986
9/16	1 7/9	.2912	.2085	.2033	.3529	.3591
1/2	2	.2600	.1853	.1801	.3147	.3199
7/16	2 2/7	.2287	.1622	.1570	.2752	.2804
2/5	2 1/2	.2100	.1482	.1430	.2518	.2570
3/8	2 2/3	.1975	.1390	.1338	.2359	.2411
1/3	3	.1766	.1235	.1183	.2098	.2150
5/16	3 1/5	.1662	.1158	.1106	1966	.2018
2/7	3 1/2	.1528	.1059	.1007	.1797	.1849
1/4	4	.1350	.0927	.0875	.1573	.1625
2/9	4 1/2	.1211	.0824	.0772	.1398	.1450
1/5	5	.1100	.0741	.0689	.1259	.1311
3/16	5 1/3	.1037	.0695	.0643	.1179	.1232
1/6	6	.0933	.0617	.0565	.1049	.1101
1/7	7	.0814	.0530	.0478	.0899	.0951
1/8	8	.0725	.0463	.0411	.0787	.0839
1/9	9	.0655	.0413	.0361	.0699	.0751
1/10	10	.0600	.0371	.0319	.0629	.0681
1/16	16	.0412	.0232	.0180	.0392	.0444

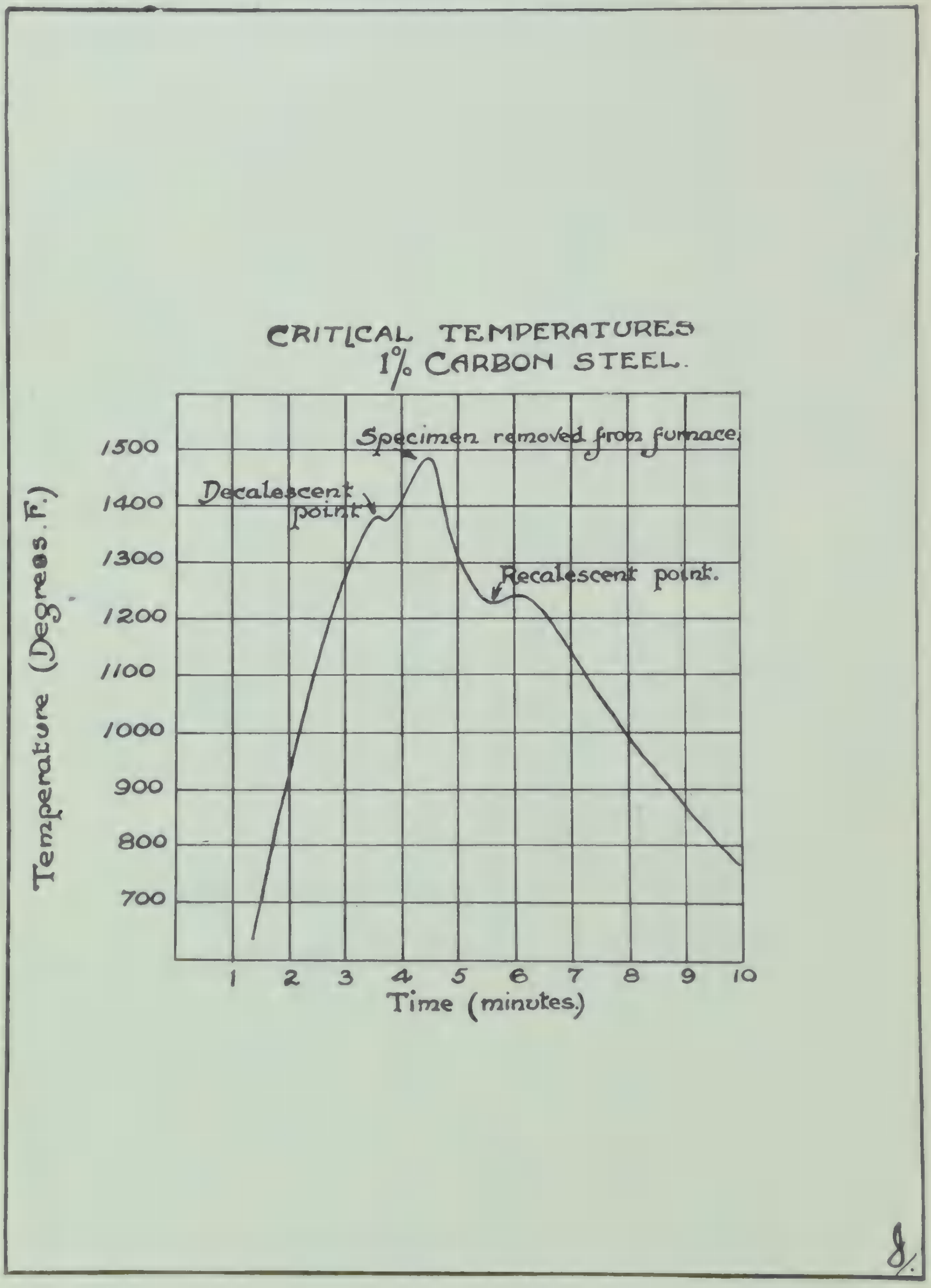
DECIMAL EQUIVALENTS OF PARTS OF AN INCH

$\frac{1}{64}$ . . .	.01563	$\frac{21}{64}$ . . .	.32813	$\frac{45}{64}$ . . .	.70313
$\frac{1}{32}$	.03125	$\frac{11}{32}$	.34375	$\frac{23}{32}$	.71875
$\frac{3}{64}$ . . .	.04688	$\frac{23}{64}$ . . .	.35938	$\frac{47}{64}$ . . .	.73438
1-16	.06250	**3-8**	.37500	**3-4**	.75000
$\frac{5}{64}$ . . .	.07813	$\frac{25}{64}$ . . .	.39063	$\frac{49}{64}$ . . .	.76563
$\frac{3}{32}$	.09375	$\frac{13}{32}$	.40625	$\frac{25}{32}$	.78125
$\frac{7}{64}$ . . .	.10938	$\frac{27}{64}$ . . .	.42188	$\frac{51}{64}$ . . .	.79688
1-8	.12500	**7-16**	.43750	**13-16**	.81250
$\frac{9}{64}$ . . .	.14063	$\frac{29}{64}$ . . .	.45313	$\frac{53}{64}$ . . .	.82813
$\frac{5}{32}$	.15625	$\frac{15}{32}$	.46875	$\frac{27}{32}$	.84375
$\frac{11}{64}$ . . .	.17188	$\frac{31}{64}$ . . .	.48438	$\frac{55}{64}$ . . .	.85938
3-16	.18750	**1-2**	.50000	**7-8**	.87500
$\frac{13}{64}$ . . .	.20313	$\frac{33}{64}$ . . .	.51563	$\frac{57}{64}$ . . .	.89063
$\frac{7}{32}$	.21875	$\frac{17}{32}$	.53125	$\frac{29}{32}$	.90625
$\frac{15}{64}$ . . .	.23438	$\frac{35}{64}$ . . .	.54688	$\frac{59}{64}$ . . .	.92188
1-4	.25000	**9-16**	.56250	**15-16**	.93750
$\frac{17}{64}$ . . .	.26563	$\frac{37}{64}$ . . .	.57813	$\frac{61}{64}$ . . .	.95313
$\frac{9}{32}$	.28125	$\frac{19}{32}$	.59375	$\frac{31}{32}$	.96875
$\frac{19}{64}$ . . .	.29688	$\frac{39}{64}$ . . .	.60938	$\frac{63}{64}$ . . .	.98438
5-16	.31250	**5-8**	.62500	**1**	1.00000
		$\frac{41}{64}$ . . .	.64063		
		$\frac{21}{32}$	.65625		
		$\frac{43}{64}$ . . .	.67188		
		11-16	.68750		

DIFFERENT STANDARDS FOR WIRE GAUGES

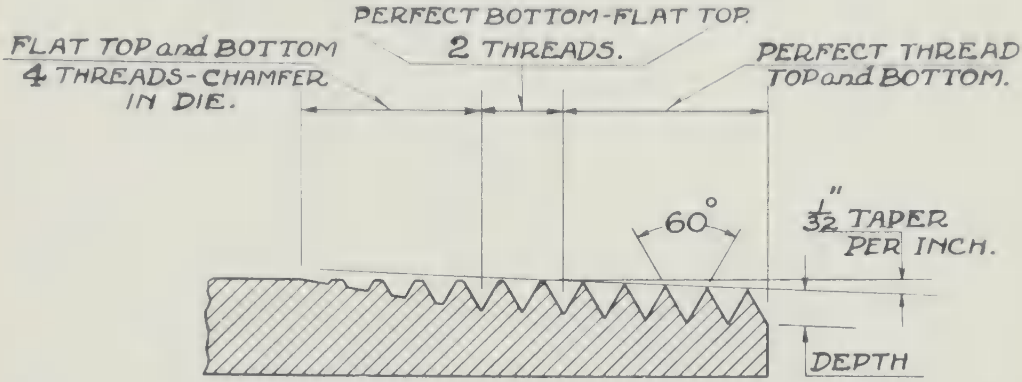
Dimensions of Sizes in Decimal Parts of an Inch

Number of Wire Gauge	American, or Brown & Sharpe (Sheet Copper and Brass)	Birmingham, or Stubs' Iron Wire (Iron Telegraph)	Steel Wire Gauge	Stubs' Steel Wire (Drill Rod)	British Imperial Wire Gauge	New American S & W Co.'s Music Wire Gauge	U. S. Standard Gauge for Sheet and Plate Iron and Steel
	......	....	......	....	....	...	....
	......	....	......	....	....	...	....
000000	......	....	......	....	.464	.004	.46875
00000	......	....	......	....	.432	.005	.4375
0000	.460	.454	.3938	....	.400	.006	.40625
000	.40964	.425	.3625	....	.372	.007	.375
00	.3648	.380	.3310	....	.348	.008	.34375
0	.32486	.340	.3065	....	.324	.009	.3125
1	.2893	.300	.2830	.227	.300	.010	.28125
2	.25763	.284	.2625	.219	.276	.011	.265625
3	.22942	.259	.2437	.212	.252	.012	.250
4	.20431	.238	.2253	.207	.232	.013	.234375
5	.18194	.220	.2070	.204	.212	.014	.21875
6	.16202	.203	.1920	.201	.192	.016	.203125
7	.14428	.180	.1770	.199	.176	.018	.1875
8	.12849	.165	.1620	.197	.160	.020	.171875
9	.11443	.148	.1483	.194	.144	.022	.15625
10	.10189	.134	.1350	.191	.128	.024	.140625
11	.090742	.120	.1205	.188	.116	.026	.125
12	.080808	109	.1055	.185	.104	.029	.109375
13	.071961	.095	.0915	.182	.092	.031	.09375
14	.064084	083	.0800	.180	.080	.033	.078125
15	057068	.072	.0720	.178	.072	.035	.0703125
16	.05082	065	.0625	.175	.064	.037	.0625
17	.045257	058	.0540	.172	.056	.039	.05625
18	.040303	.049	.0475	.168	.048	.041	.050
19	.03589	042	.0410	.164	.040	.043	.04375
20	.031961	.035	.0348	.161	.036	.045	.0375
21	.028462	.032	.03175	.157	.032	.047	.034375
22	.025347	.028	.0286	.155	.028	.049	.03125
23	.022571	.025	.0258	.153	.024	.051	.028125
24	.0201	.022	.0230	.151	.022	.055	.025
25	.0179	.020	.0204	.148	.020	.059	.021875
26	.01594	.018	.0181	.146	.018	.063	.01875
27	.014195	016	.0173	.143	.0164	.067	.0171875
28	.012641	.014	.0162	.139	.0149	.071	.015625
29	.011257	.013	.0150	.134	.0136	075	.0140625
30	.010025	.012	.0140	.127	.0124	.080	.0125
31	.008928	.010	.0132	.120	.0116	.085	.0109375
32	.00795	.009	.0128	.115	.0108	.090	.01015625
33	.00708	.008	.0118	.112	.0100	.095	.009375
34	.006304	.007	.0104	.110	.0092	....	.00859375
35	.005614	.005	.0095	.108	.0084	...	.0078125
36	.005	.004	.0090	.106	.0076	...	.00703125
37	.004453	....	....	.103	.0068	....	.006640625
38	.003965	....	.....	.101	.0060	....	.00625
39	.003531	....	.....	.099	.0052	...	....
40	.003144	....	.....	.097	.0048	....	....



CRITICAL TEMPERATURES
1% CARBON STEEL.
Specimen removed from furnace.
Decalescent point
Recalescent point.
1500
1400
1300
1200
1100
1000
900
800
700
Temperature (Degrees F.)
1
2
3
4
5
6
7
8
9
10
Time (minutes.)

Briggs's Standard Pipe Thread



Pipe Diameters			Depth of Thread	No. Threads per Inch	Length of Thread Inches	No. Turns Pipe Screws into Fitting by Hand	No. Turns to be Made with Wrench	Inches Pipe Screws into Fitting	No. of Turns Pipe Screws into Fitting
Nominal Size	Actual Inside	Actual Outside							
⅛	0.270	0.405	.029	27	.412	4	1.13	0.19	5.13
¼	0.364	0.540	.044	18	.624	4	1.22	0.29	5.22
⅜	0.494	0.675	.044	18	.634	4	1.40	0.30	5.40
½	0.623	0.840	.057	14	.818	4	1.46	0.40	5.46
¾	0.824	1.050	057	14	.828	4	1.60	0.51	5.78
1	1.048	1.315	.069	11½	1.03	4½	1.37	0.55	6.21
1¼	1.380	1.660	.069	11½	1.06	5	1.21	0.58	6.33
1½	1.610	1.900	.069	11½	1.07	5	1.33	0.89	6.67
2	2.067	2.375	.069	11½	1.10	5	1.67	0.95	7.12
2½	2.468	2.875	.100	8	1.64	5	2.12	1.00	7.60
3	3.067	3.500	.100	8	1.70	5	2.60	1.05	8.00
3½	3.548	4.000	.100	8	1.75	5	3.00	1.05	8.40
4	4.026	4.500	.100	8	1.80	5½	2.90	1.10	8.80
4½	4.508	5.000	.100	8	1.85	5½	3.30	1.10	8.80
5	5.045	5.563	.100	8	1.91	5½	3.78	1.16	9.28
6	6.065	6.625	.100	8	2.01	6	4.08	1.26	10.08
7	7.023	7.625	.100	8	2.11	7	3.88	1.36	10.88
8	7.981	8.625	.100	8	2.21	8	3.68	1.46	11.68
9	8.941	9.625	.100	8	2.32	9	3.56	1.57	12.56
10	10.020	10.750	.100	8	2.43	10	3.44	1.68	13.44

ALLOWANCES FOR FITS

NOMINAL DIAMETERS	0 to $\frac{1}{2}''$	$\frac{9}{16}''$ to $1''$	$1\frac{1}{16}''$ to $2''$	$2\frac{1}{16}''$ to $3''$	$3\frac{1}{16}''$ to $4''$	$4\frac{1}{16}''$ to $5''$
ALLOWANCES FOR FORCED FITS.						
HIGH LIMIT.	+0·0010	+0·0020	+0·0040	+0·0060	+0·0080	+0·0100
LOW "	+0·0005	+0·0015	+0.0030	+0.0045	+0.0060	+0.0080
TOLERANCE	0·0005	0·0005	0.0010	0.0015	0.0020	0.0020
ALLOWANCES FOR DRIVING FITS.						
HIGH LIMIT	+0·0005	+0·0010	+0.0015	+0.0025	+0.0030	+0.0035
LOW "	+0·0002	+0·0007	+0.0010	+0.0015	+0.0020	+0.0025
TOLERANCE	0·0003	0·0003	0.0005	0.0010	0.0010	0.0010
ALLOWANCES FOR PUSH FITS.						
HIGH LIMIT.	-0·0002	-0·0002	-0.0002	-0.0005	-0.0005	-0.0005
LOW "	-0·0007	-0·0007	-0.0007	-0.0010	-0.0010	-0.0010
TOLERANCE	0·0005	0·0005	0·0005	0·0005	0·0005	0·0005
ALLOWANCES FOR RUNNING FITS.						
HIGH LIMIT	-0·0010	-0·0012	-0·0017	-0·0020	-0·0025	-0·0030
LOW "	-0·0020	-0·0027	-0·0035	-0·0042	-0·0050	-0·0057
TOLERANCE	0·0010	0·0015	0·0018	0·0022	0·0025	0·0027
HIGH LIMIT	-0·0007	-0·0010	-0·0012	-0·0015	-0·0020	-0·0022
LOW "	-0·0012	-0·0020	-0·0025	-0·0030	-0·0035	-0·0040
TOLERANCE	0·0005	0·0010	0·0013	0·0015	0·0015	0·0018
HIGH LIMIT.	-0·0005	-0·0007	-0·0007	-0·0010	-0·0010	-0·0012
LOW "	-0·0007	-0·0012	-0·0015	-0·0020	-0·0022	-0·0025
TOLERANCE	0·0002	0·0005	0·0008	0·0010	0·0012	0·0013

MACHINE SCREW THREADS

Tap Size	Threads Per In.	Tap Drill No.	Body Drill Dec.	Body Drill
2	56	48	.093	42
3	48	44	.104	37
4	36	41	.120	31
5	40	36	.136	29
6	32	33	.144	27
7	32	30	.157	22
8	32	28	.169	18
9	32	24	.185	13
10	32	20	.196	9
10	24	23	.196	9
11	24	19	.213	3
12	24	15	.228	1
14	24	6	.250	¼
14	20	10	.250	¼

INDEX

MACHINE SHOP PRACTICE
BOOK II